ArcSWAT

ArcGIS Interface for Soil and Water Assessment Tool (SWAT)

http://www.brc.tamus.edu/swat

Getnet Dubale Betrie

Yunqing Xuan

Ann van Griensven

UNESCO-IHE, Delft, Netherlands

R. Srinivasan

r-srinivasan@tamu.edu

Blackland Research and Extension Center and Spatial Sciences Laboratory

Texas Agricultural Experiment Station

Texas A&M University

Table of Contents

Introduction

The ***Soil and Water Assessment Tool (SWAT)*** is a physically-based continuous-event hydrologic model developed to predict the impact of land management practices on water, sediment, and agricultural chemical yields in large, complex watersheds with varying soils, land use, and management conditions over long periods of time. For simulation, a watershed is subdivided into a number of homogenous subbasins (hydrologic response units or HRUs) having unique soil and land use properties. The input information for each subbasin is grouped into categories of weather; unique areas of land cover, soil, and management within the subbasin; ponds/reservoirs; groundwater; and the main channel or reach, draining the subbasin. The loading and movement of runoff, sediment, nutrient and pesticide loadings to the main channel in each subbasin is simulated considering the effect of several physical processes that influence the hydrology. For a detailed description of the capabilities of the ***SWAT***, refer to *Soil and Water Assessment Tool User's Manual, Version 2000* (Neitsch et al., 2002), published by the Agricultural Research Service and the Texas Agricultural Experiment Station, Temple, Texas. The manual can also be downloaded from the ***SWAT*** Web site (www.brc.tamus.edu/swat/swatdoc.html#new).

Objectives

The objectives of this exercise are to (1) setup a SWAT project and (2) familiarize with the capabilities of SWAT.

Create a Project

ArcSWAT extension of ArcGIS 9.1 or 9.2 creates an ArcMap project file that contains links to your retrieved data and incorporates all customized GIS functions into your ArcMap project file. The project file contains a customized ArcMap Graphical User Interface (GUI) including menus, buttons, and tools. The major steps on how to create a SWAT project under then ArcMap environment are introduced below:

Step 1. Start ArcMap. Under the **Tools** menu of ArcMap View, click the **Extensions** button. You will see an extension entitled "SWAT Project Manager" and "SWAT Watershed delineator" under the extensions list (Figure 1). Turn on these two extensions.

Step 2. Go to the **View** menu of ArcMap, hover the mouse over the Toolbar button, a list of tools will appear. Click the ArcSWAT tool, the main interface of ArcSWAT will open (Figure 2).

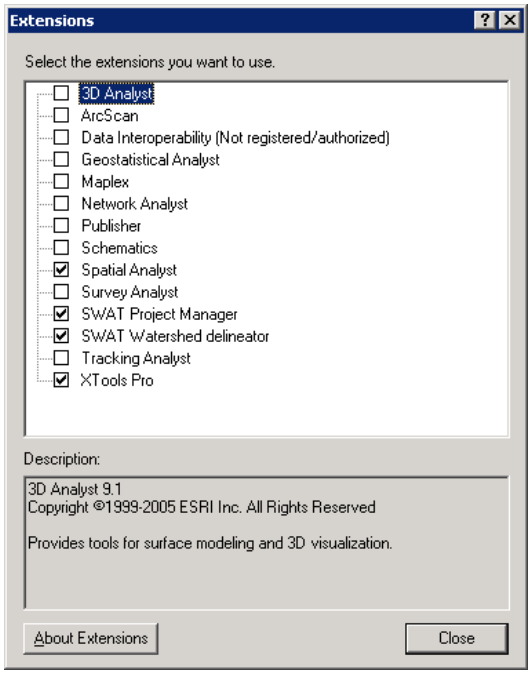


Figure 1 Extensions of ArcMap

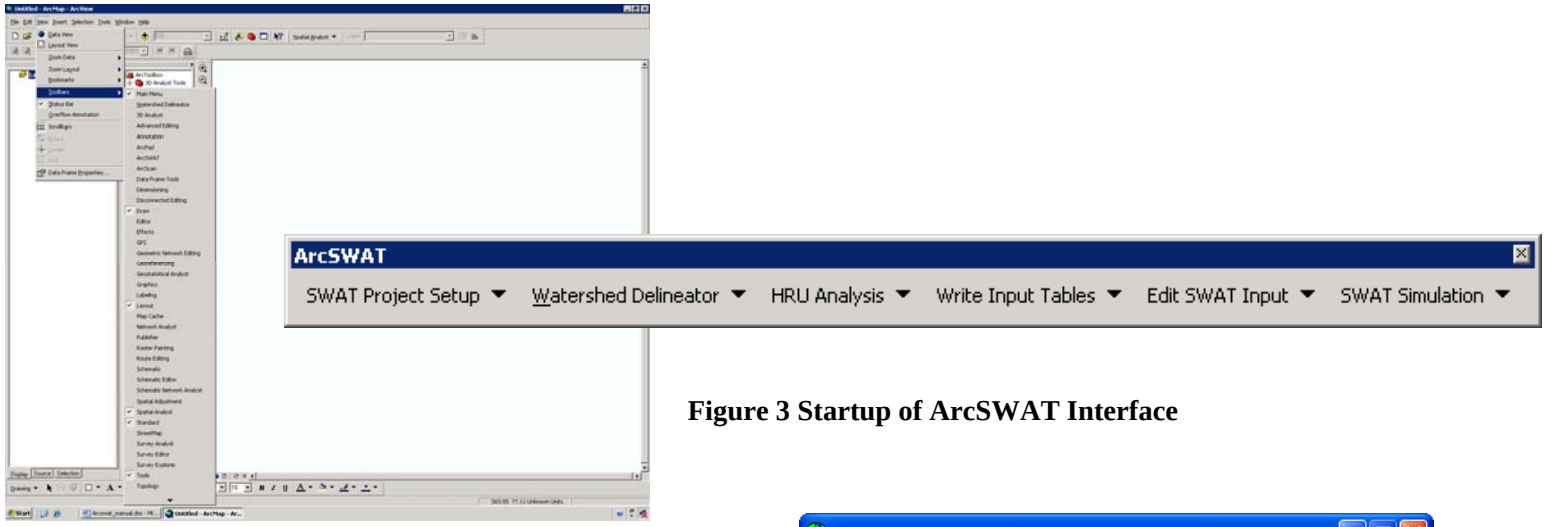


Figure 3 Startup of ArcSWAT Interface

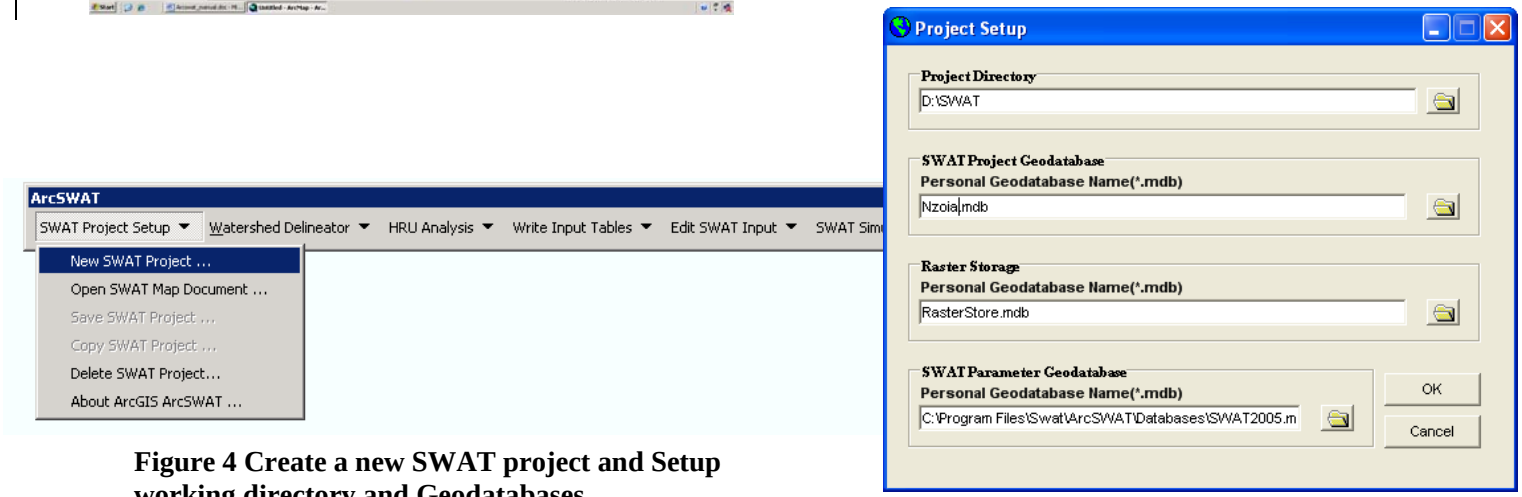


Figure 4 Create a new SWAT project and Setup working directory and Geodatabases

Step 3. Create a new SWAT project and Setup working directory and Geodatabases

- Click the **New SWAT Project** button under then **SWAT Project Setup** menu. In the prompted dialog entitled "**Project Set Up**" (Figure 3), Select a **Project Directory** for current project**.** The names of **SWAT Project Geodatabase** and **Raster Storage Geodatabase** are initialized automatically. These two Geodatabases will be created under the **Project Directory.**
- The **SWAT Parameter Geodatabase** stores the parameters that are needed for SWAT model run. The directory of this Geodatabase is provided by the interface, users should not change this value.
- Click **OK** after you setup a project. Then a new SWAT project will be created under the **Project Directory (**Figure 4**).** The SWAT project includes two folders, two geodatabases, and one <Project Directory>.mxd file. The < Project Directory >.mxd file is the file that is currently in use by the user.

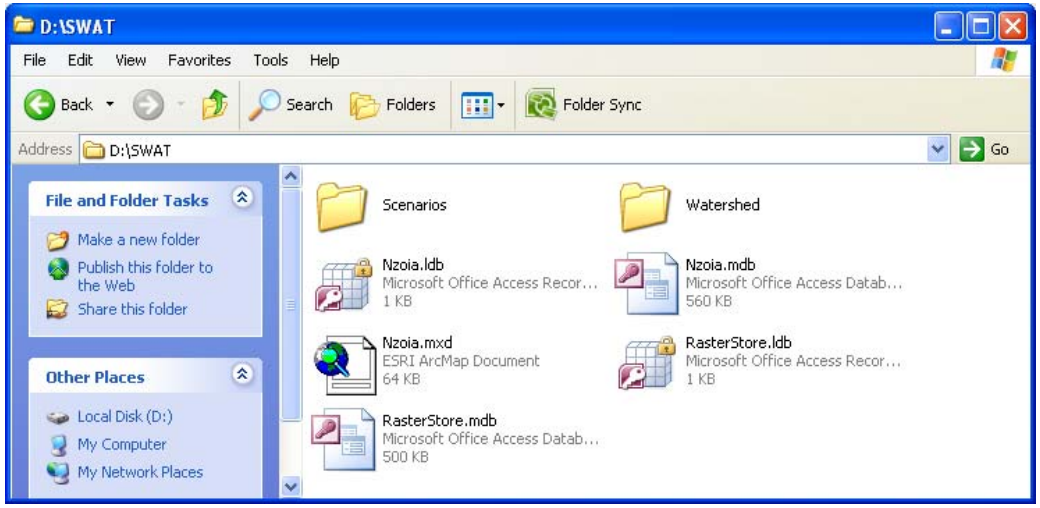


Figure 4 The structure of SWAT project directory

Watershed Delineation

After setup of a new SWAT project, activate the **Automatic Watershed Delineation** button under the **Watershed Delineator** menu by clicking this button. The **Watershed Delineation** tool will appear (Figure 5). The tool's functions are divided into five sections, namely: DEM setup, Stream Definition, Outlet and Inlet Definition, Watershed Outlet(s) Selection and Definition. This tool is used to create wateshed delineations using a combination of DEM, digitized network (RF1 or RF3, NHD or User defined), and other user inputs. The detailed procedures on how to use the **Watershed Delineation** tool are introduced in the following sections:

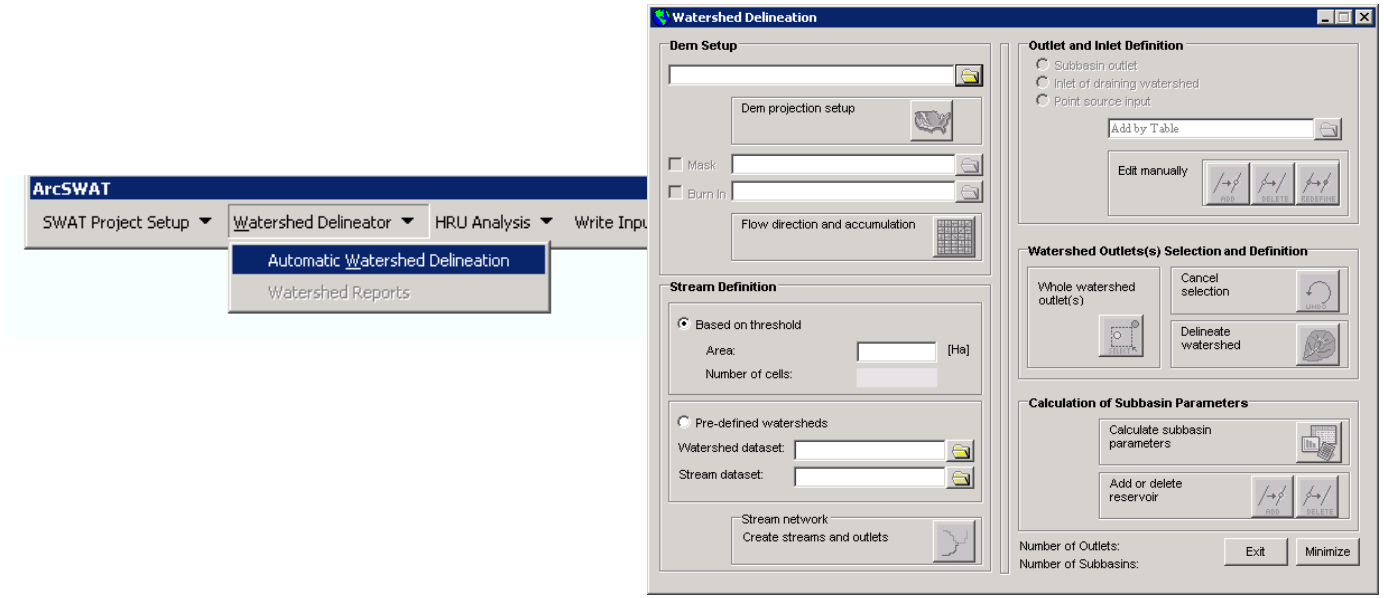


Figure 5 Watershed Delineation main dialog

Step 1: Add DEM Grid

1. Click the  icon to add the DEM grid to be setup for modeling with SWAT. A dialog box (Figure 6) with options **Select from Watershed View** and **Load DEM grid from disk** is opened. Choose **Load DEM grid from disk** and select the **dem** grid located in your disk (the directory is "../Nzoia-data/DEM" for this example) and click **ADD**. The selected dem will be resaved as **<Project Directory>\RasterStore.mdb\SourceDem, which is loaded into the ArcMap.**

   Note: the selected dem file must have an effective projection. If not, an error dialog will prompt.

2. After loading **dem**, the  button will be activated. Using this button, the user can check the projection information of the **dem (**Figure 7**)**. The DEM properties box lets you verify that the DEM map properties are correct and make any needed changes. The DEM properties should correctly define the grid size and units. The user can change the Z unit to reflect the real situation.

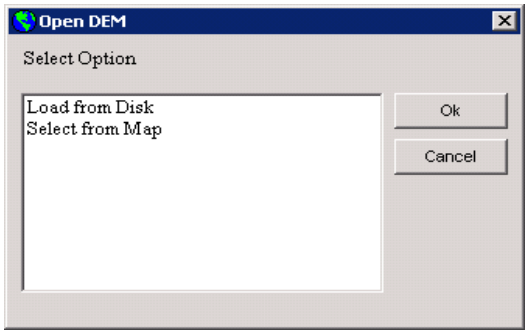


Figure 6 Projection properties of DEM

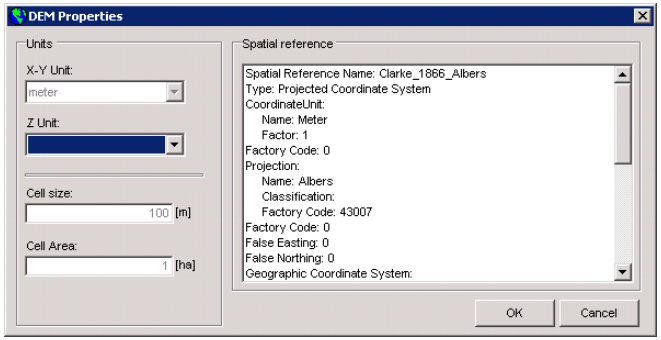


Figure 7 Projection properties of DEM

Step 2: Focusing on Watershed Area

The interface allows users to import or create a grid map that masks out a part of the DEM grid and/or a shape map that defines the stream network. These maps are not required but might speed the processing time for the GIS operations.

1. Click  beside **Mask**, then three options are available to specify the watershed area of interest (Figure 8): **Load mask grid from disk**, **manually delineate** and **Select boundary theme from basins view**. Select the **Load mask grid from disk** option and click **OK**. Select the **mask** grid located in your disk, and click **ADD.** The selected dem will be resaved as **<Project Directory>\RasterStore.mdb\Mask,** which is loaded into the ArcMap (Figure 10).
2. If you selected the **Manually Delineate** option, then a toolset will appear and allow you draw polygon mask (Figure 9). A toolbar with three buttons, **DRAW**, **EDIT VERTEX** and **DELETE** appears. These tools provide the capability similar to drawing and editing a polygon theme in ArcMap.
   Click on the **DRAW** button to manually trace the boundary of the watershed area of interest using the mouse. Double click the mouse left button to complete the polygon. **EDIT Vertex** will allow you to adjust the boundary of the mask.
   The **DELETE** Button will allow you to select the entire polygon for deletion using the "Delete" function in your Keyboard. Click **APPLY** after you are satisfied with the manually drawn masking area. This will create a masking grid (**MASK**) and add the theme to the Watershed View.

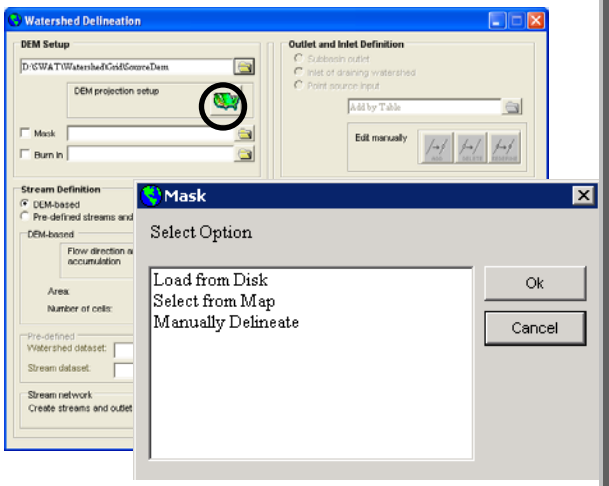


Figure 8 Options to define Mask

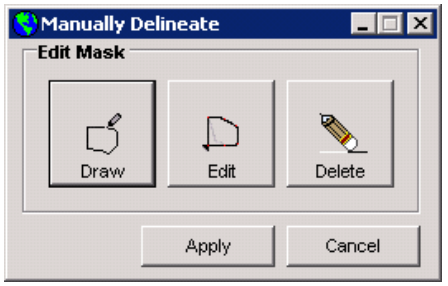


Figure 9 Toolset for Manually Mask Delineation

Note:
1. If a mask grid already exists, you can select Load mask grid from the disk option. The grid will be added to the Basins view and used for delineation.
2. A polygon theme already in the Basins View can be selected as a mask, using the Select boundary theme from the Basins view option.

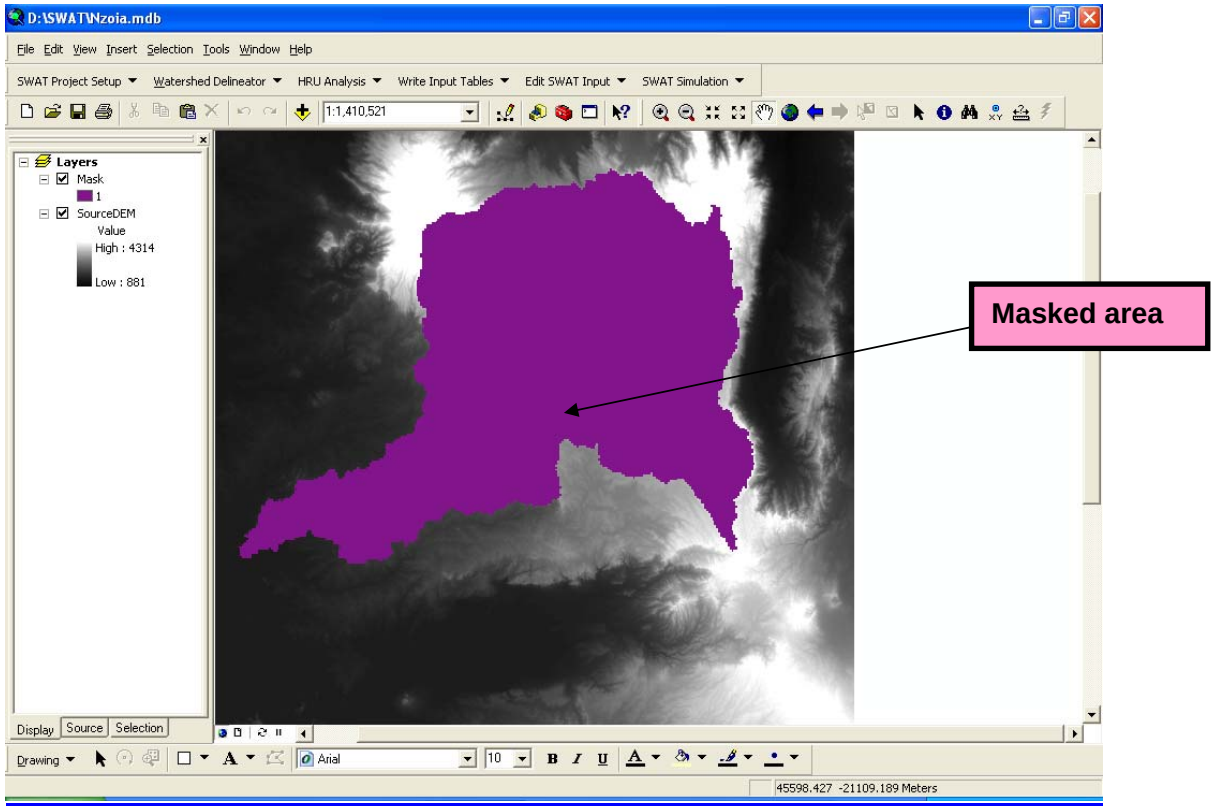


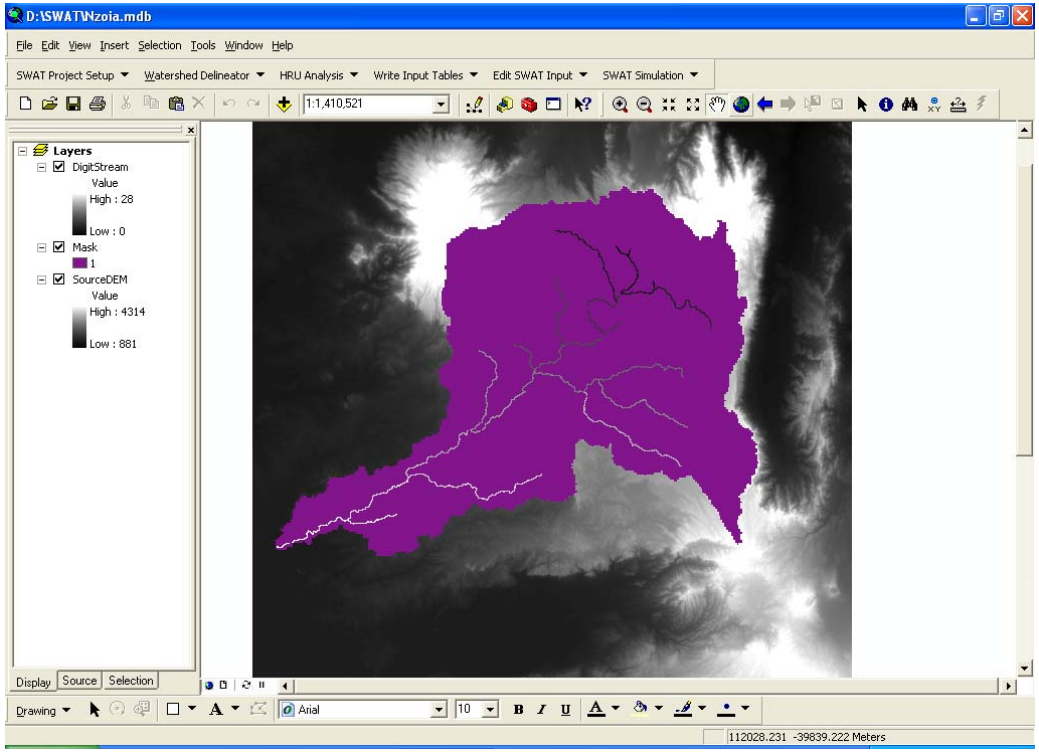
Figure 10 DEM grid added to ArcMap View with a mask area

Note:

- You can exit the main Watershed Delineation Dialog anytime and open it again by selecting **AUTOMATIC DELINEATION** from the WATERSHED **DELINEATOR** menu.

Step 3: Burning in a stream network

A stream network theme such as Reach File (V1 or V3) or National Hydrography Dataset (NHD) can be superimposed onto the DEM to define the location of the stream network.



Note:

- NHD is an enhanced stream network at the scale of 1:100,000. It is based on USGS Digital Line Graph (DLG) hydrography data integrated with reach-related information from the EPA River Reach File version 3 (RF3).

Step 4: Stream Definition

(Note: For the Stream Definition function of the Watershed Delineator, There are two ways to define the watershed and stream network. In this section, the method based threshold area will be introduced, while another method based on pre-defined watershed will be introduced in Appendix I.)

1. In order to use the threshold method to delineate the watershed and stream network, the **Flow Direction and Accumulation** needs to be calculated by clicking the  button. Stream definition defines both the stream network and subbasin outlets. A minimum, maximum, and suggested sub watershed area (in hectares) is shown in the drainage area box (Figure 11). You have the option of changing the size of the subbasins within the specified range of values. This function plays an important role in determining the detail of the stream network and the size and number of subbasins created. The threshold area defines the drainage area required to form the beginning of a stream.

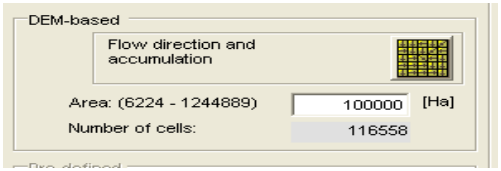


Figure 11 Threshold area for stream and subbasin definition

2. After setting the threshold value of subbasin, then the user can delineate the stream network and outlets through clicking the  button. The drainage network and stream juncture points, used to define subbasin outlets, are displayed on the DEM map grid (**Figure 12**).

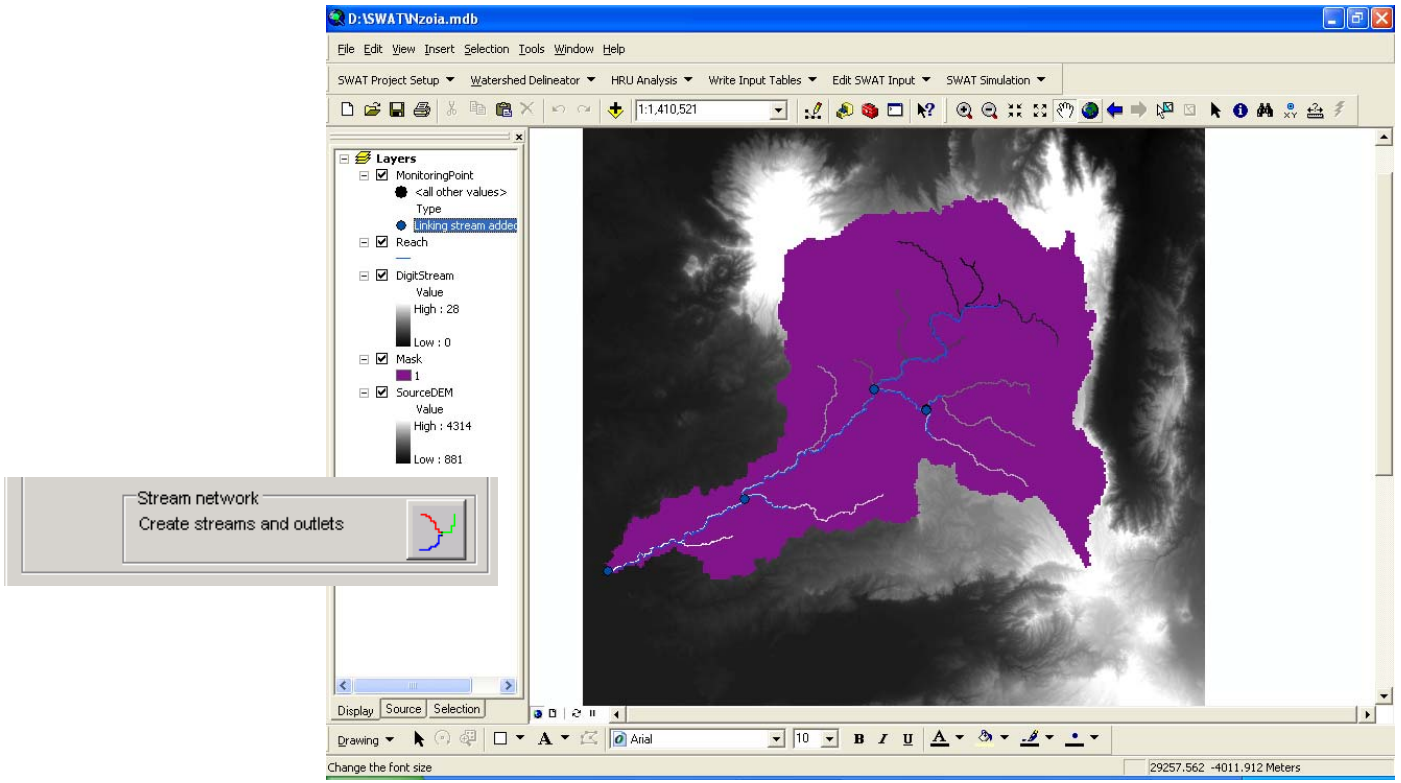


Figure 12 Drainage network and subbasin outlets

3. Next step is **Outlet and Inlet Definition** (Figure 13). Before proceeding, you have a number of options: (1) change the threshold area and rerun the stream and outlet definition routine, (2) add outlet points by importing a table that contains the locations, (3) add outlet points manually, and (4) remove outlet points. Assuming the outlet and stream definition to be acceptable proceed to inlet definition for the study area. Inlets represent any point source loading into the study area or the inlets of drainage into the watershed from an upstream area.

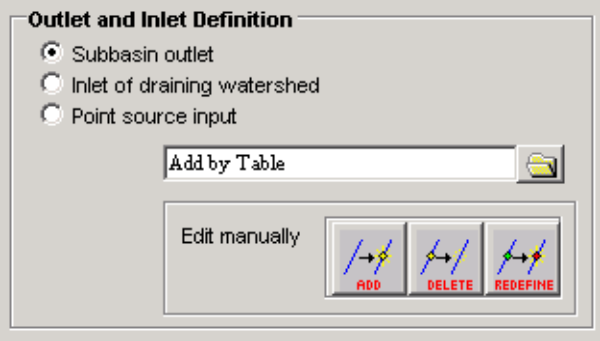


Figure 13 Options for defining Outlet and Inlet

Note

- By specifying the threshold area, we define the stream network for networking.
- It means that a minimum number of cells are required to start delineating the stream.
- The minimum threshold area is for the entire watershed, not for each sub watersheds that are going to be delineated.
- The suggested area given in this window is the average are that could be used.

Step 5: Main Watershed Outlet(s) Selection and Definition

In this step the users will select one or more outlet locations to define the boundary of the main watershed.

1. Click on the **SELECT** button  to choose the watershed outlet. Draw a box covering the desired outlet locations will set the main **Watershed Outlets**. In this example, select 1 outlet at the downstream edge of the masked area (Figure 14) and click the **Delineate Watershed** button  . Select **YES** in the following dialog to continue with the delineation of main watershed and subbasins. A prompt box will appear to announce completion of the watershed and subbasin delineation.
2. The delineated watershed with subbasins will be added to the View. If the delineation is not satisfactory or if the user wants to select a different outlet for the watershed, click on the **Cancel Selection** button and repeat.
3. Click on the **Calculate Subbasin Parameters button**  to estimate the subbasin parameters. This function calculates basic watershed characteristics from the DEM and sub-watershed themes. It also assigns the necessary subbasin identification. The results of the calculations are stored as additional fields in the streams and subbasins theme database files. Click **OK** to completion of watershed delineation dialog box. Figure 15 shows the delineated watershed with subbasins.
4. Open the Reach or Watershed attribute tables to view the calculated characteristics.

** By holding the **SHIFT** key in your keyboard you can select more than one outlet. This feature allows adjacent watershed to be simulated at the same time using SWAT. Do not select an outlet at the upstream of another outlet. At least one outlet must be selected for delineation.

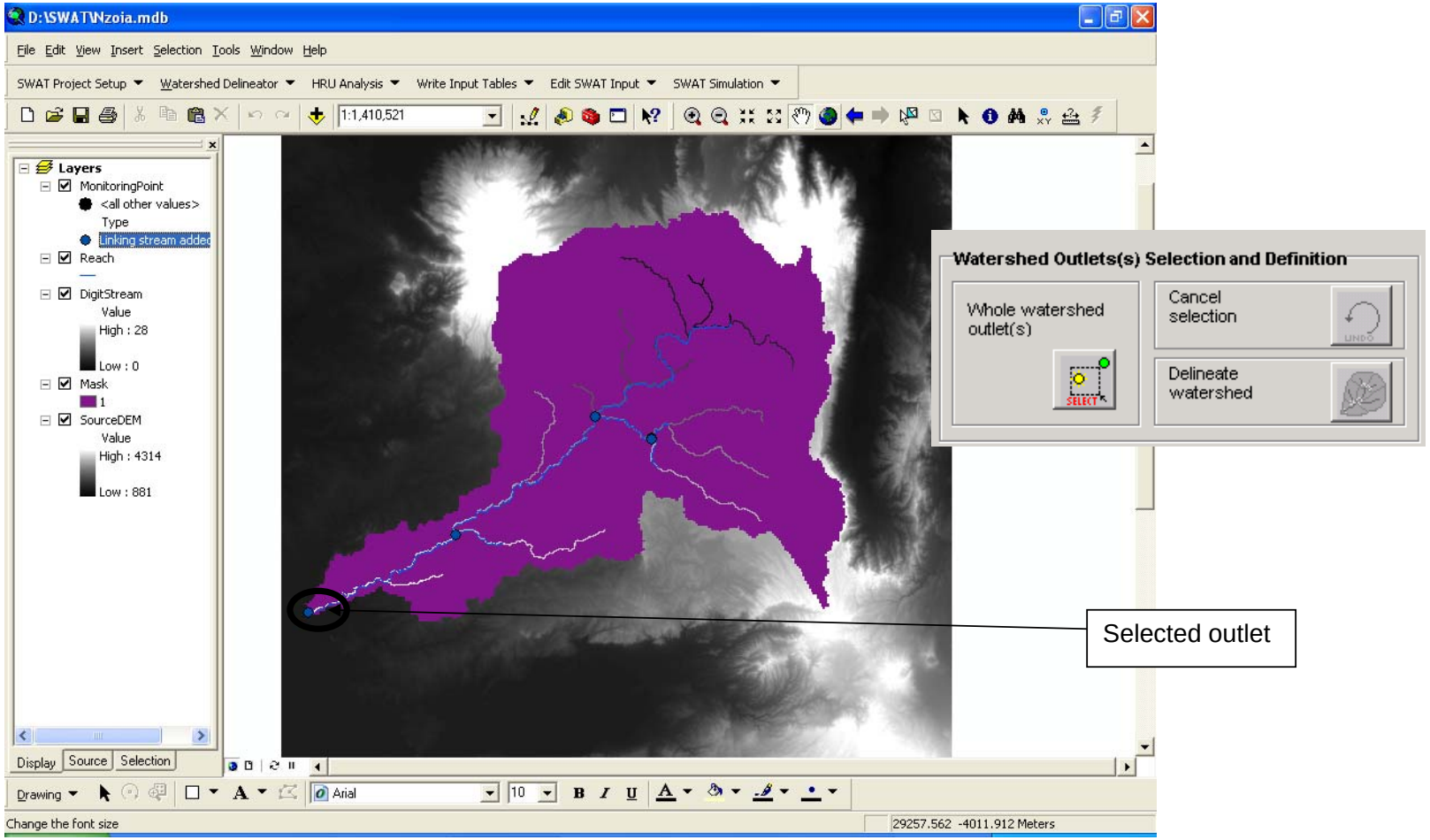


Figure 14 Main watershed outlet selection

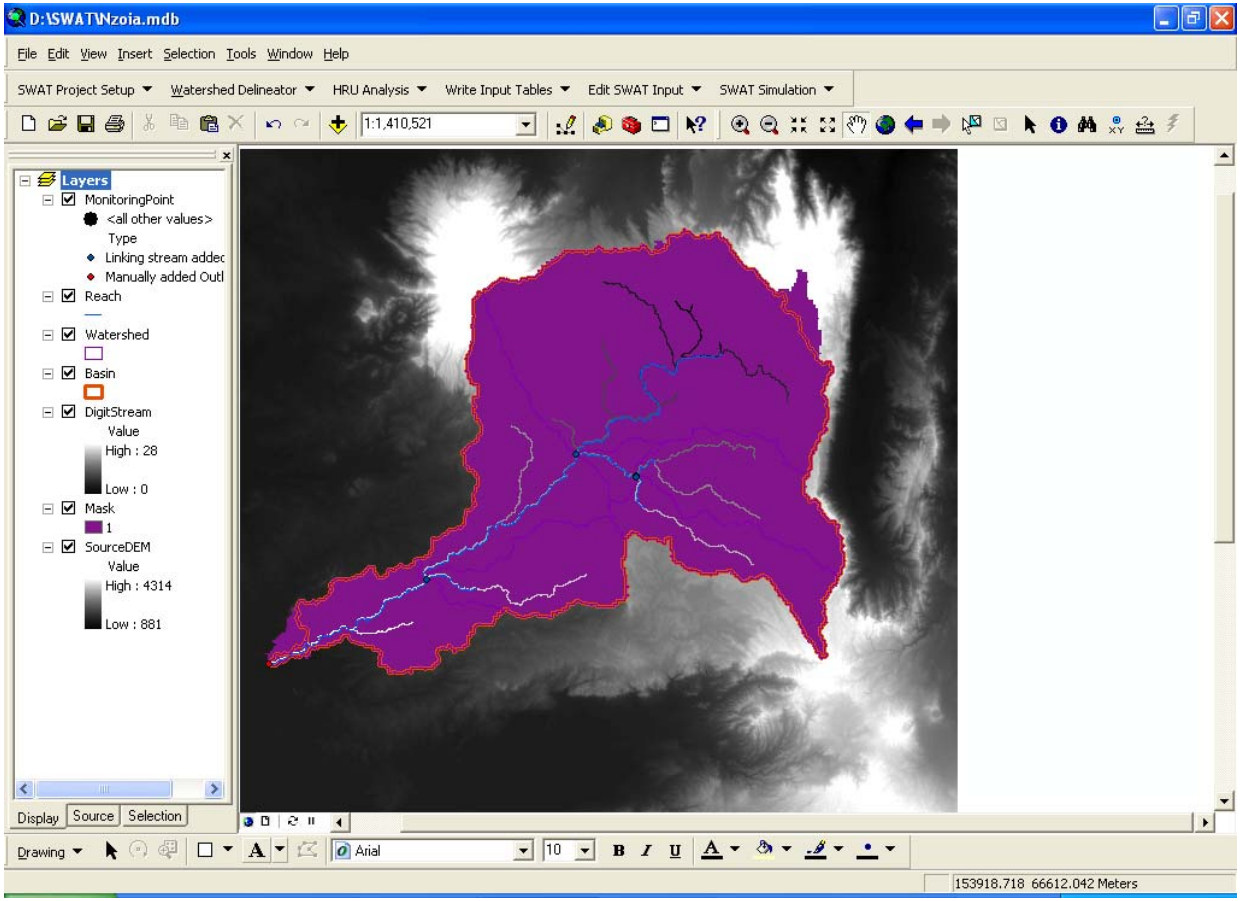


Figure 15 Delineated watershed and subbasins

Step 6: Reservoirs

The user can add or remove reservoirs to refine the delineation process. The procedure is similar to the process of manually adding or removing an outlet**. <u>In this exercise, we will not focus on adding or removing a reservoir.</u>** However the general procedure is given below for reference.

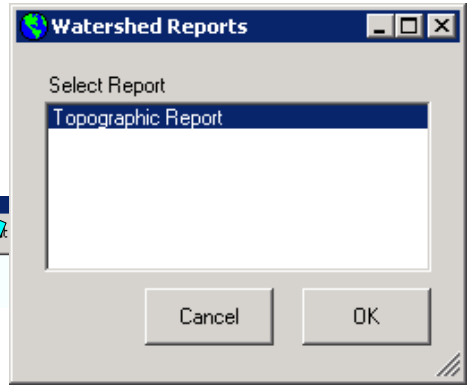
1. To add a reservoir, click on the **ADD** button and with the cross appearing as mouse pointer, click on the subbasin where the reservoir will be added. The reservoir will be placed at the outlet of the subbasin. A prompt box will be displayed asking for verification of the reservoir placement. If the wrong subbasin is listed, click **No** and repeat step 7(1). If the correct subbasin is listed, click **Yes**.
2. To remove a reservoir click on the **REMOVE** button. Draw a square around the reservoir to be removed by holding the left mouse button and moving the mouse. A prompt box will be displayed asking for verification of the reservoir removal. If the wrong subbasin is listed, click **No** and repeat step 7(2). If the correct subbasin is listed, click **Yes**.

Step 8: Exit the Watershed Delineation Dialog

1. Click **EXIT** in the watershed delineation main dialog

Step 9: View Topographic Report

Open the **Topographic Reports** through click the Watershed Reports Button under the **Watershed Delineator** menu from **Reports** of the project windows. You can view the area and percentage wise distribution of elevation for the entire watershed and each subbasin.



Land use, Soil and Slope Definition

The **Land Use, Soil and Slope Definition** option in the **HRU Analysis** menu allows the user to specify the land use, soil and slope themes that will be used for modeling using SWAT and NPSM. These themes are then used to determine the hydrologic response unit (HRU) distribution in each sub-watershed.

Both NPSM and SWAT require land use data to determine the area of each land category to be simulated within each subbasin. In addition to land use information, SWAT relies on soil data to determine the range of hydrologic characteristics found within each subbasin. **Land Use, Soil and Slope Definition** option guides the user through the process of specifying the data to be used in the simulation and of ensuring that those data are in the appropriate format. In particular, the option allows the user to select land use or soil data that are in either shape or grid format. Shapefiles are automatically converted to grid, the format required by ArcGIS to calculate land use and soil distributions within the subbasins of interest. Select the **Land Use / Soil / Slope Definition** option from the **HRU Analysis** menu. The **Land Use / Soil / Slope Definition** dialog box (Figure 16) will open. The detailed procedures on how to use the functions contained in this dialog were introduced below:

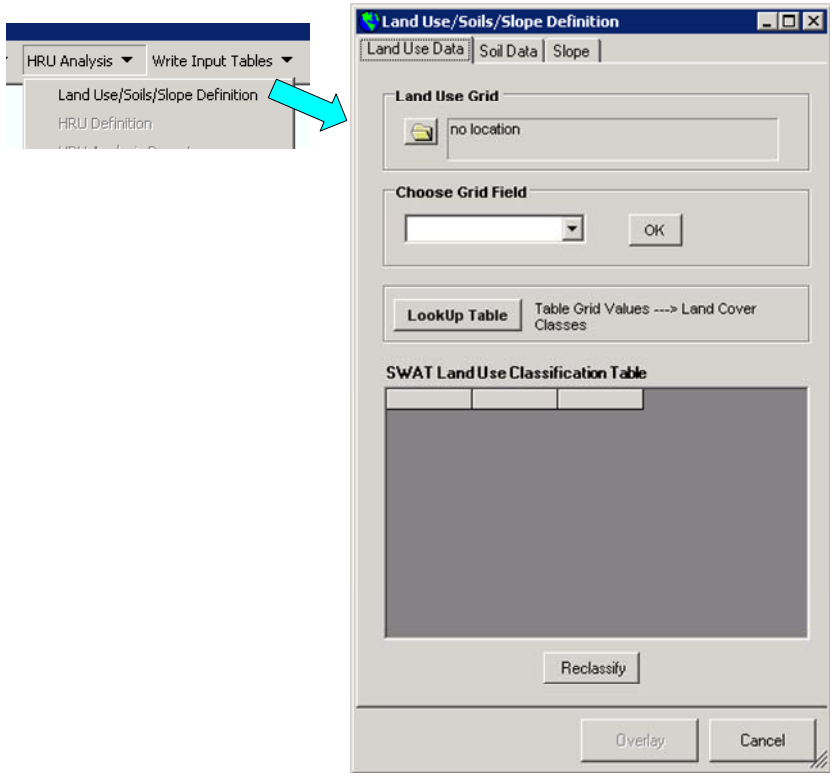


Figure 16 Dialog for Land Use / Soil / Slope Definition

Step 1: Define Land use theme

1. Select the land use data layer by clicking on the open file folder button  next to "**Land Use Grid**."
2. A "Set the LandUse Grid" dialog box will appear (Figure 17). You will have the option to "**Select Land use layer(s) from the Map"** or "**Load Land Use dataset(s) from disk"**. Select the **Load Land Use dataset(s) from disk** option and click **Open**. Click **Yes** for the projection information dialog box.
3. Select the **Landuse** grid file in the **work directory** and click **Select**. A message box will indicate the successful loading of landuse theme.

Figure 17 Dialog for options of selecting landuse dat

4. After loading the Landuse file into the map, choose the grid field which will be used as index to define different landuse types. In this example, the "**Value**" field is selected. Click **OK**, then a table titled "**SWAT LandUse Classification Table**" will be created automatically by the interface (Figure 18). The first column contains the unique values in the **Grid Field** chosen above. The second column contains the area of each type of landuse. And the third column contains the landuse names in the SWAT database corresponding to each index value.

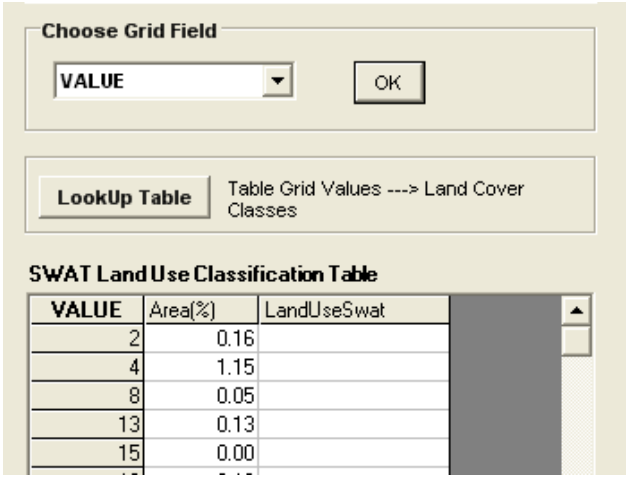
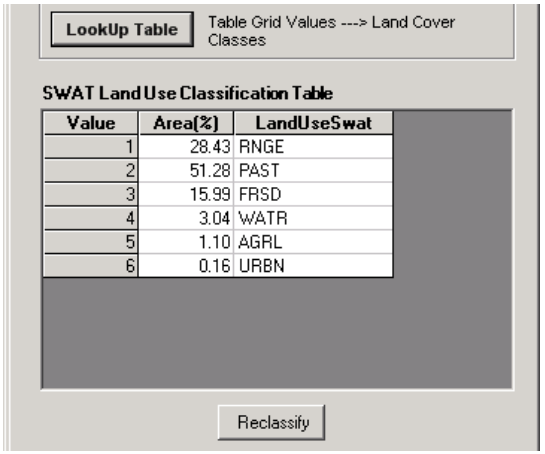


Figure 19 SWAT LandUse Classification Table

5. In order to fill correct values in the third column, the land use grid codes must be assigned a land cover/plant description. You may import a look-up table or manually assign a land cover/plant code. The interface includes tables that convert the USGS land use/land cover classification codes to SWAT land cover/plant codes. If the land use grid being used is classified by an alternate method, you must create a look-up table or enter the information manually.
6. Select the land use look-up table by clicking on the open "**Look-up Table**" button next to **"Table Grid Values –> Land cover classes**." A "Land Cover Lookup Table" dialog box will appear (Figure 19).
7. When the user has land use data and the corresponding four character land use code, the "**User Table**" option should be selected.
8. Select User defined lookup table in .txt or .dbf format (the ASCII .txt lookup table was shown in(Figure 21), then the SWAT LandUse Classification Table will be populated (Figure 20).
9. Click **Reclassify** button. Then Landuse map will be reclassified according the Lookup table. The reclassified landuse map is shown in Figure 22.



Accessing “User Table” from the Landuse and soil definition dialog box:

- The user defined table has to be created in the format as shown in figure below:

ASCII (.txt) Table Format

An example land use look up file is:

```
"Value","Landuse"
1,RNGE
2,PAST
3,FRSD
4,WATR
5,AGRL
6,URBN
```

Figure 20 Populated SWAT LandUse Classification Table

Figure 21 ASCII (.txt) table format of lookup table

Note:

1. To manually create a look-up table, double click on the “LandUseSwat” field next to the first category number in the dialog. A dialog box will appear listing the two database files from which a SWAT land type may be selected: Land Cover/Plant and Urban. Select the desired database file by clicking on it. Click **O**K. A dialog box will appear listing the available SWAT land cover codes or the available SWAT urban land type codes. Select the desired code from the list and click ok. Repeat this procedure for all the values in the grid.
2. If you do not find the desired land cover in the database, you will have to add the land cover class to the database too.

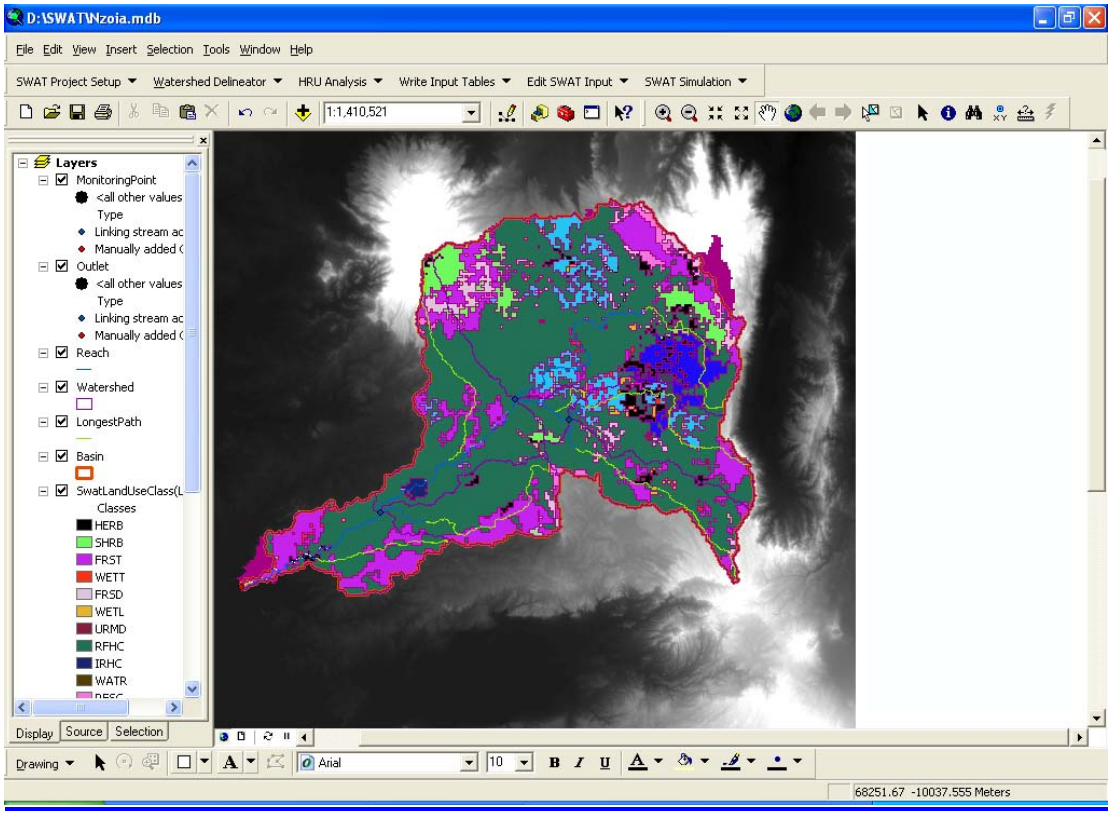


Figure 22 Reclassified land use grid

Step 2: Specify soils theme

1. Select the soils data layer by clicking on the open file folder button  under "**Soil Grid**" in the "**Soil Data**" tab (Figure 23).
2. A "Select Soils Data" dialog box appears. You have the option to **Select Soil layer(s) from the Map** or **Load Soils dataset(s) from disk**. Select the Soil map from the disk and load it to the map. A message box will indicate the successful loading of the soil grid theme.
3. After loading the **Soil** file into the map, choose the field which will be used as the index to define different soil types. In this example, the "**Value**" field is selected. Click **OK**, then a table titled "**SWAT Soil Classification Table**" will be created automatically by the interface. The first column contains the unique values in the **Grid Field** chosen above. The second column contains the area of each soil type. And the third column contains the soil names in the SWAT database corresponding to each index value, which need to be filled through a Lookup table.

The soil map grid must be linked to the U.S. soils database (provided with the interface) or to the User Soils (customized) database. Under "Options," click the button beside the method used to link the soils map grids to the soils data. There are five options available:

- **Stmuid**: State STATSGO polygon number, dominant soil

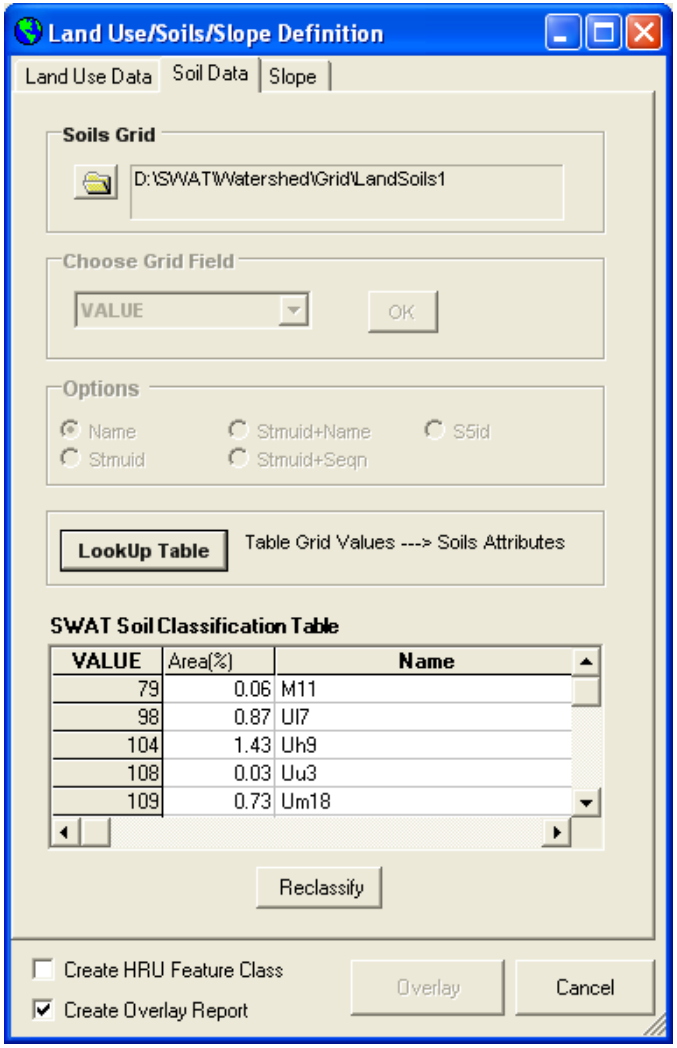


Figure 23 Interface for setting the soil data

phase

- **S5id**: Soils5ID number for USDA soil series data
- **Name**: Name of soil in User Soils database
- **Stmuid + Seqn**: State STATSGO polygon number and sequence number of soil phase
- **Name + Stmuid**: State STATSGO polygon number and soil series name

4. Select **Name,** then load look up values for the soil grid file and click the **Reclassify** button for soils grid. The reclassified soils grid (Figure 24) is shown in the map.

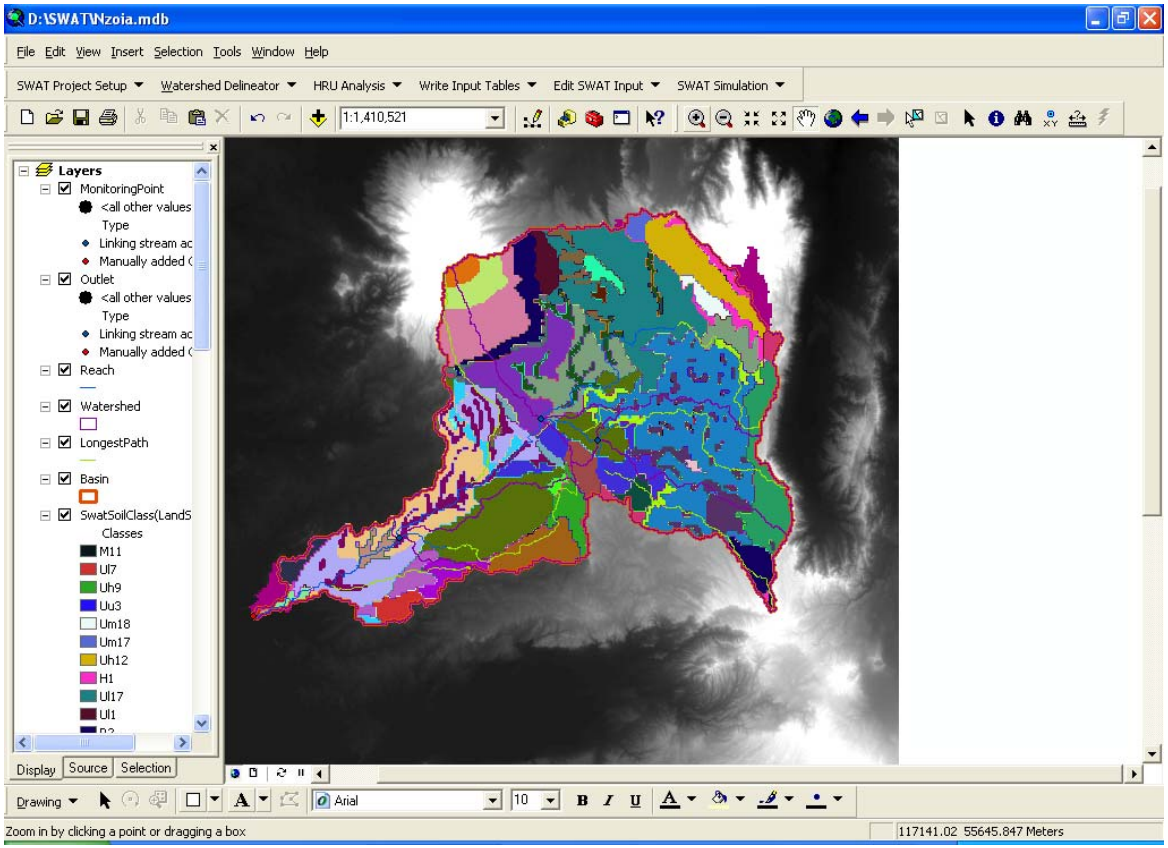


Figure 24 Soils grid theme reclassified by Soil names

Note:

1. You can manually reclass the soils grid using a procedure similar to the manual reclassification of land use grid.
2. User defined lookup tables in database or text format can also added by click on the "OPEN FOLDER"

Step 3: Specify slope theme

Slope is an important factor that determines the water, sediment and nutrients movement. In this new version interface, a new function is added to take slope into account to delineate HRU. The general procedures for defining a slope theme will be introduced below.

1. Click the "**Slope"** tab, then the interface for defining slope theme appear (Figure 25). There are two options for slope discretization: "**Single Slope**" and "**Multiple Slope**" The "**Single Slope**" option denotes that the mean value of slope will be used for for the whole watershed, whilethe "**Multiple Slope**" option will allow slope be classified into several level.

   If the "**Multiple Slope**" option was selected, the user need to define the "**Number of Slope Classes",** and the range (**Lower Limit and Upper Limit**) of each slope class. At last the slope classification results will be stored in the "**SWAT Slope Classification Table**". Click "Reclassify", the classified slope layer will be added to the map.

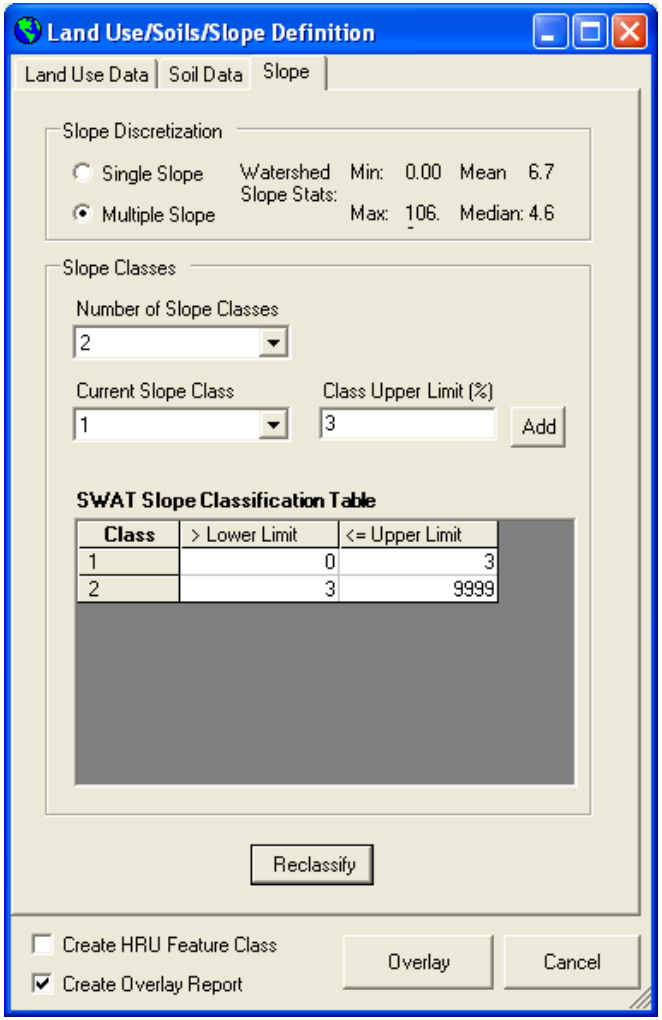


Figure 25 Populated SWAT LandUse Classification Table

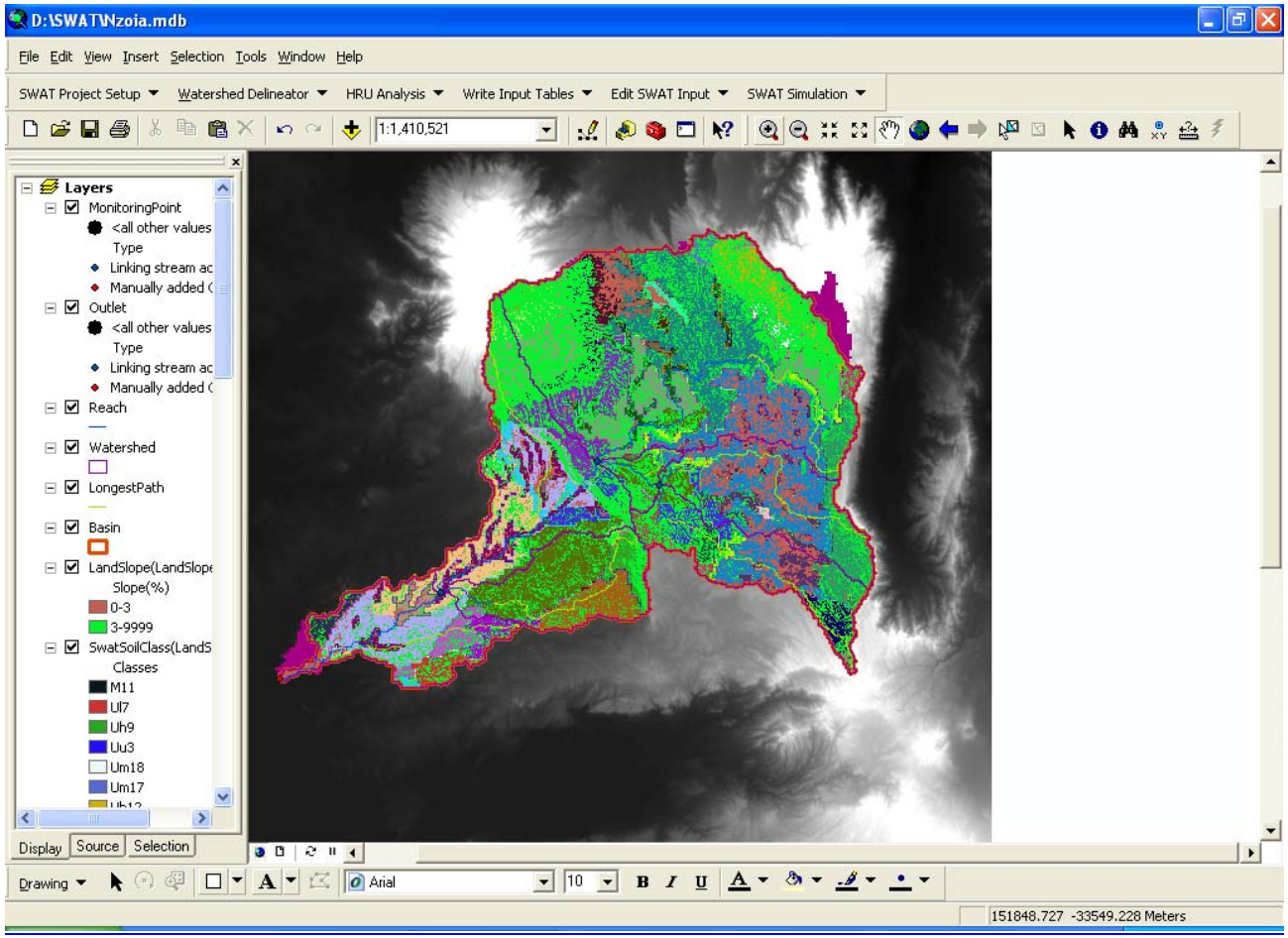


Figure 26 Slope grid theme reclassified by user defined criteria

Step 4: Overlay land use, soil and slope layers

When the land use, soil and slope layers have all been classified, the **OVERLAY** button will be activated. Click this button; all the three grid layers will be overlaid. When the overlay process is completed, an "Info" message box will appear. Click **OK** to proceed the delineation of hydrologic response units (HRUs). Figure 27 shows the units with unique combination of landuse, soil, and slope within each subbasin. This layer will be used later in defining HRU.

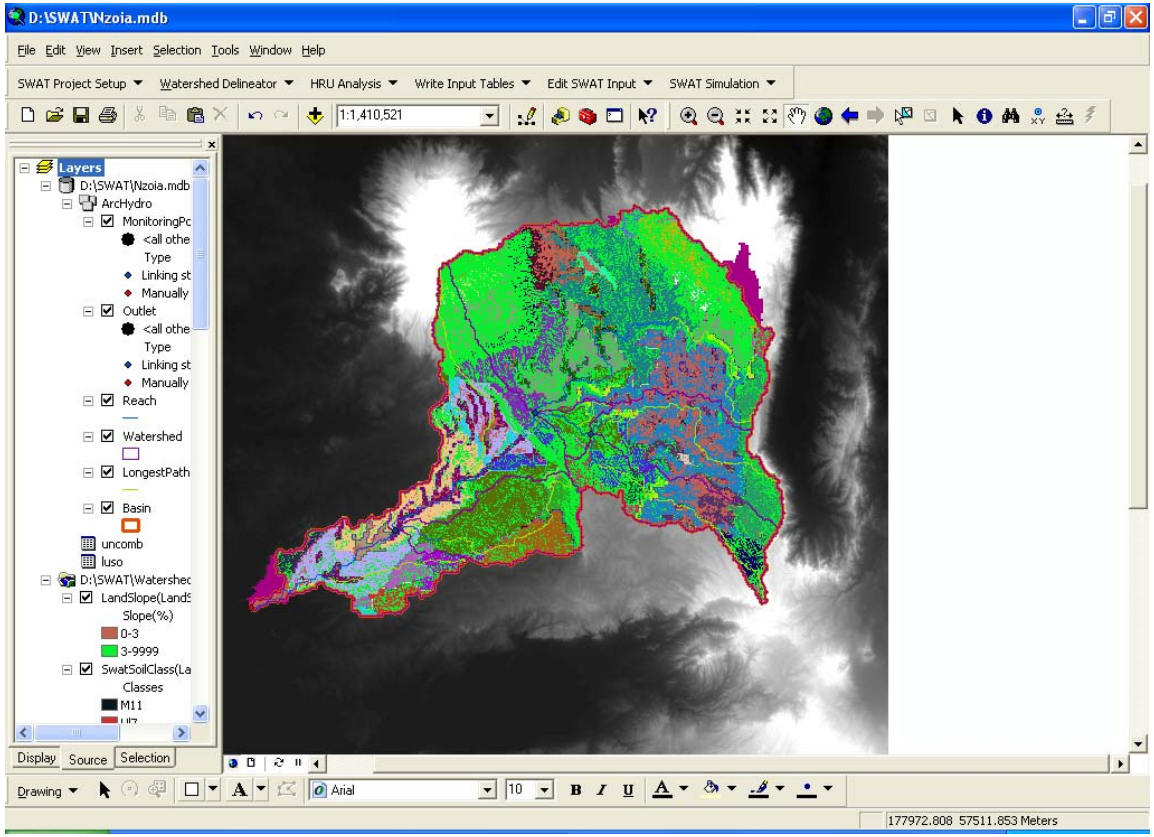


Figure 27 Shape file layer of the units with unique combination of landuse, soil and slope within each subbasin

Step 5: View land use, soil and slope distribution report

View the area and percentage distribution of each land use and soil class within each subbasin by clicking the **HRU Analysis Reports** button under the **HRU analysis** menu.

Hydrologic Response Unit Definition

Before executing SWAT, the distribution of hydrologic response units (HRUs) within the watershed must be determined based on the land use, soil and slope layers specified in the previous step. The interface allows the user to specify criteria to be used in determining the HRU distribution. One or more unique land use/soil/slope combination(s) (hydrologic response units or HRUs) can be created for each subbasin.

Subdividing the watershed into areas having unique land use, soil and slope combinations enables the model to reflect differences in evapotranspiration for various crops and soils. Runoff is predicted separately for each HRU and routed to obtain the total runoff for the watershed. This increases accuracy and gives a much better physical description of the water balance.

Click the **HRU Definition** button under the **HRU Analysis** menu. This will open a "**HRU Definition**" dialog (Figure 28). The user has two options in determining the HRU distribution: assign a single HRU to each subbasin or assign multiple HRUs to each subbasin. If a single HRU per subbasin is selected, the HRU is determined by the dominant land use/soil/slope combination within each subbasin. If multiple HRUs are selected, the user may specify the several criteria for the land use, soil and slope data

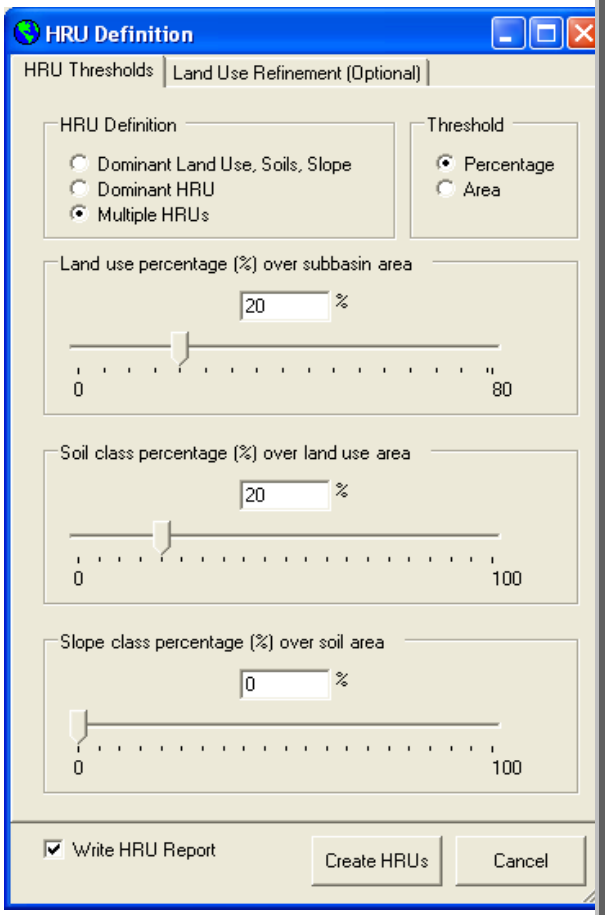


Figure 28 HRU definition dialog

that will be used to determine the number and type of HRUs in each subbasin. The procedures on how to use this tool are introduced below:

Step 1: Hydrologic Response Unit Definition

1. Select the **Multiple Hydrologic Response Units** option. Modify the threshold land use/soil/slope area percentage (%) over subbasin area to define how detailed the watershed will be represented. Select the desired threshold values for soil, land use and slope (for example 10% for land use/soil/slope)**.**

2. Now you can click the "**Create HRUs**" button to delineate HRU distribution. Also you can go to the Land Use Refinement tap to specify more detailed criteria. There are two ways to refine the definition of HRU. The first one is to split one land use type into two or several sub land use types. For example the AGRL land use type can be split into two sub types: AGRL (50%) and AGRR (50%). The other option is to set one land use type exempt, which mean that this type of land use will exempt of the area threshold value set in previous steps. In this example, the settings of these two functions are shown in Figure 29.

3. After creating HRUs, the distribution report for the HRUs can open through clicking the **HRU Analysis Reports** button under the **HRU**

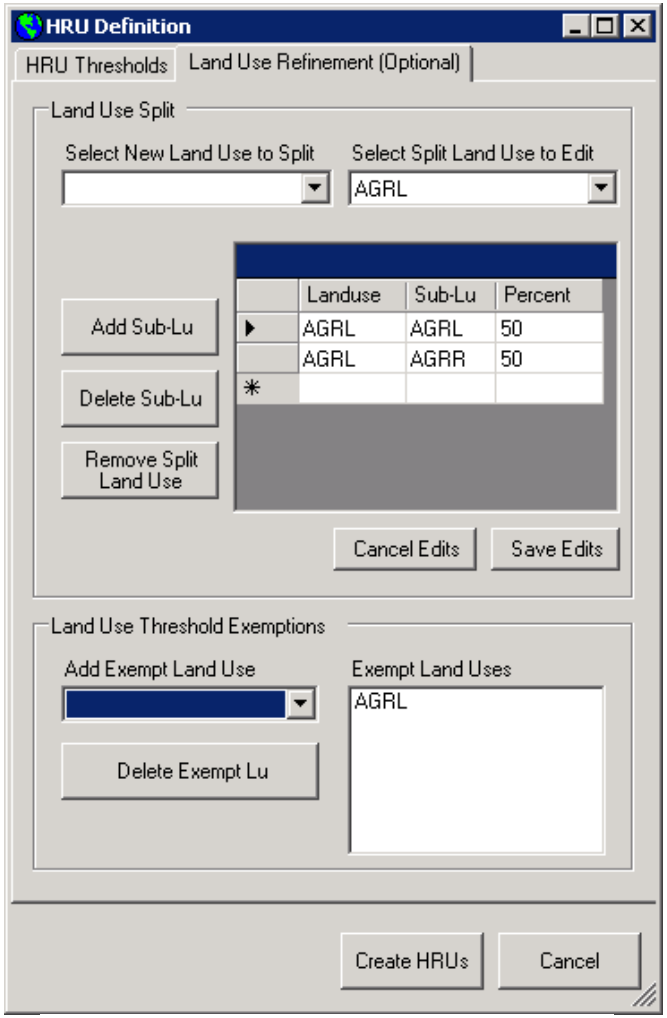


Figure 29 Interface for Land use refinement

Analysis menu.

4. If the distribution is not satisfactory, repeat the preceding steps, altering the land use and soil sensitivities, until a satisfactory distribution is obtained.

Note for HRU distribution

- Selecting Multiple Hydrologic response units option allows us to eliminate minor land uses in each subbasin.
- For example, if we set the threshold for Landuse (%) over subbasin area to 15% landuses that occupy less that 15% of subbasin area would be eliminated and the HRU will be created for landuses that occupy greater than 15% of the subbasin area.
- The same holds for Soil and slope layer.

Soil and Water Assessment Tool (SWAT)

The following are the key procedures necessary for modeling using SWAT.

- Create SWAT project
- Delineate the designated watershed for modeling
- Define land use/soil/slope data grids
- Determine the distribution of HRUs based on the land use and soil data
- Define rainfall, temperature and other weather data
- Write the SWAT input files- requires access to data on soil, weather, land cover, plant growth, fertilizer and pesticide use, tillage, and urban activities.
- Edit the input files – if necessary
- Setup and run SWAT – requires information on simulation period, PET estimation method and other options
- ***View SWAT Output***

Now we have completed the first three procedures. In this tutorial we will concentrate on preparing the rest of the input data for SWAT, running the model, and viewing the output from the model.

Note

- Spatial analyst is the main tool that will be used in SWAT. Without this, SWAT simply can't be used.
- General info about SWAT: All SWAT input and output are in Metric units (MKS)

Write Input Tables for SWAT

This menu contains functions to build database files that include information needed to generate default input for the SWAT model. The commands on the menu need to be implemented only once for a project. However, if the user modifies the HRU distribution after building the input database files, these commands must be reprocessed again.

Step 1: Define Weather data

1. Select the **Weather Stations** button under the **Write Input Tables** menu. A "Weather Data Definition" dialog is opened (Figure 30). This dialog will allow the user to define the input data for rainfall, temperature and other weather data. For weather data, you have the option of simulating the data in the model or to read from data tables. If no observed weather data is available, then information can be simulated using a weather generator. The weather generator data must be defined before you can continue to define the other data, like precipitation and temperature.

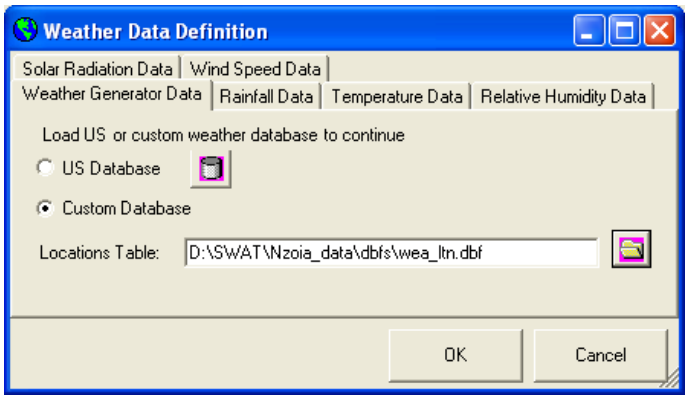


Figure 30 Weather Data definition dialog

2. Select the "Custom Database" option for Weather Simulation Data and select the ../Nzoia-data/dbfs/wea_ltn.dbf file
3. Under the "Rainfall data" tab, select the "Raingages" option for rainfall data. Browse to the **Work Directory** and choose the file **rain-ltn.dbf** and click **Add.**
4. Under the "Temperature data" tab, select the "Climate Stations" radio button option for temperature data. Browse to the **Work Directory** and choose the file **tmp-ltn.dbf** and click **Add.**

5. After selecting the rainfall, temperature and weather generator data, click **OK** to generate the SWAT weather input data files. The locations of weather generator, rainfall and temperature gages will be displayed in the map view (Figure 31).
6. A message box will indicate successful generation of SWAT weather input database.

At this point you have the option to generate all the input data files using the **WRITE ALL** option under the **INPUT** menu or generate each input file separately. The input files needed are:

- Watershed Configuration file (.fig)
- Soil data (.sol)
- Weather generator data (.wgn)
- General HRU data (.sub)
- Soil chemical input (.chm)
- Stream water quality input (.swq)
- Pond input (.pnd)
- Management Input (.mgt)
- Main channel data (.rte)
- Ground water data (.gw)
- Water use data (.wus)

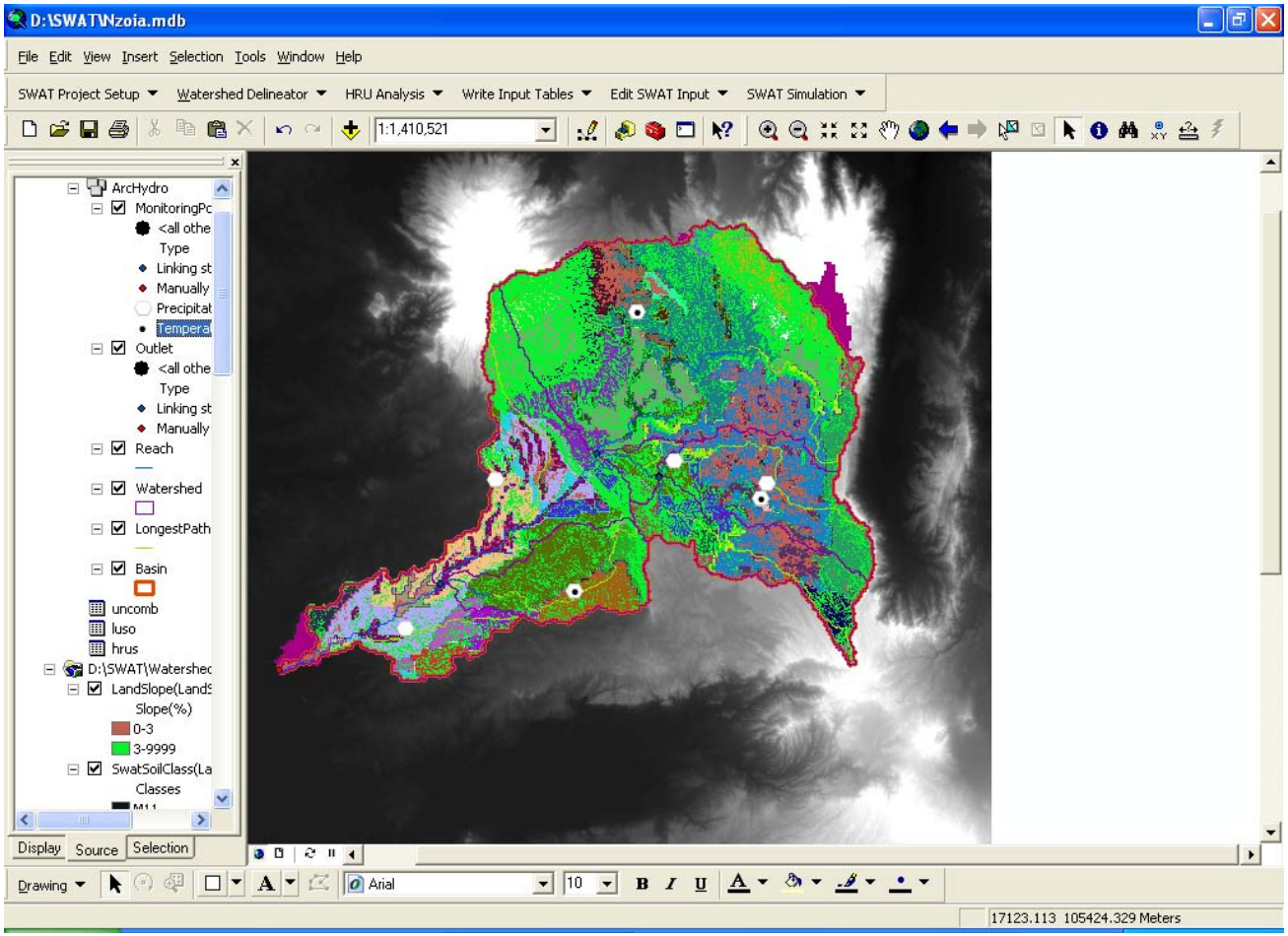


Figure 31 Location of weather stations

Step 2: Write All (Figure 32)

1. Select the **Write All** option under the **Write Input Tables** menu. A prompt appears to verify if you want to continue with the writing of all input files. Click **Yes** to continue.
2. A "Current Status of Input Data" dialog appears. This window shows the list of input files to be generated and the status of input file generation.
3. For subbasin and main channel input files a prompt asks whether to change the default Manning's n value. Click **No** to use 0.014.
4. For the management input file generation a message prompt verifies if the US weather database is sufficient to estimate the Plant Heat Units. Click **Yes** if the study area is within US.
5. A message box indicates the successful completion of input files generation.

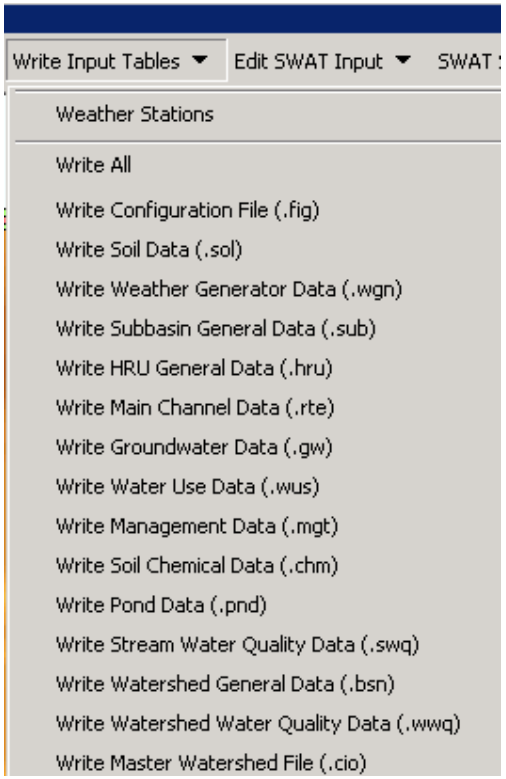


Figure 32 Write all input files status dialog

Edit SWAT Input

The commands listed under the **Edit SWAT Input** menu bring up dialog boxes that allow you to alter default SWAT input data. The **Edit SWAT Input** menu can be used to make input modifications during the model calibration process. In this exercise **you are not required** to edit any input information.

Howeve,r a general procedure is given to familiarize you with the SWAT input files and editing capabilities in ArcSWAT.

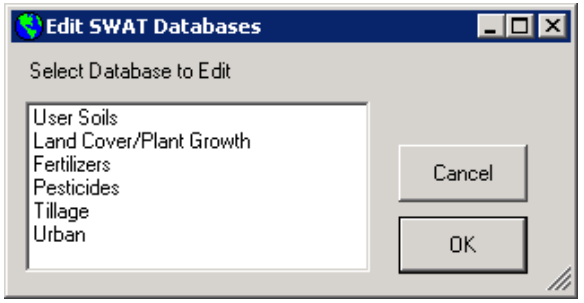


Figure 33 List of SWAT databases

Step 1: Editing Databases

1. Select the **Edit Databases** button under the **Edit SWAT Input** menu. You will be given a list of options to choose the databases to be edited (Figure 33).

Step 1-1: Edit Soils database (Figure 34)

1. Select **User Soils** option, and click **OK**.
2. "**User Soils Edit**" dialog box with a list of abbreviated soil names appears. Click on a soil name to edit the entire soil profile data or individual soil layer information.
3. You can also add new soil into the database by clicking on the **Add New** button in the bottom of the "Add and Edit User Soil" dialog box.
4. Click **EXIT** after completion of editing the database. A prompt box will give you the option to save or ignore the changes made to soils database.

By using a procedure similar to editing soils database, the **Database** option under the **Edit Input** menu you can edit or add information to the weather, land cover/plant growth, fertilizer, pesticide, tillage, and urban area databases.

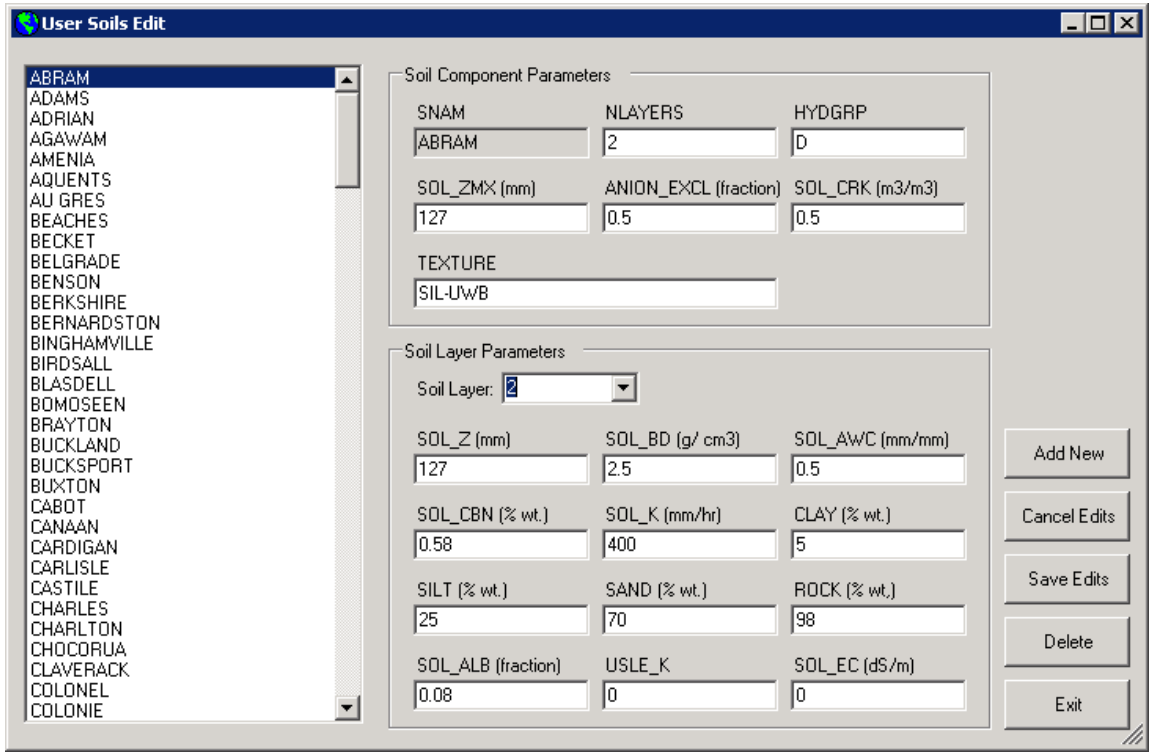


Figure 34 19 Edit soils database dialog

Note:

- Moving the mouse pointer near an object (text box, radio button etc.,) in any of the edit input dialog box will display a short description of the parameter contained in the object.

Step 2: Edit Point Discharge Inputs

1. Select the **Point Sources Discharges** option under the **Edit SWAT Input** menu. "Edit Point Source Inputs" dialog box (Figure 35) with a list of subbasins containing point discharges will appear.
2. Click on the subbasin number whose point discharge database needs to be edited. A "Point Discharges Data" dialog box appears with the list of attributes of the point data. The dialog box allows the input of point source data in one of four formats: constant daily loadings, average annual loadings, average monthly loadings, and daily loadings.
3. Choose a format by clicking the button next to the format to be used. The default point source data format option is constant daily loadings. If you select this format, you will have the option of either inputing average daily flow (m^3/s), sediment loading (tons), and organic N, organic P, NO_3, mineral P loadings (all in kg), three conservative metals, and 2 categories of bacteria or load PCS data directly. If you select the "Annual Records" option you will be prompted to load the data from disk by clicking on the open folder button or from PCS by clicking on the Load PCS button. If you select the "Monthly Records" or "Daily Records" option you will be prompted to load data from the disk. Click **OK** to complete the editing of point discharges database for a subbasin.

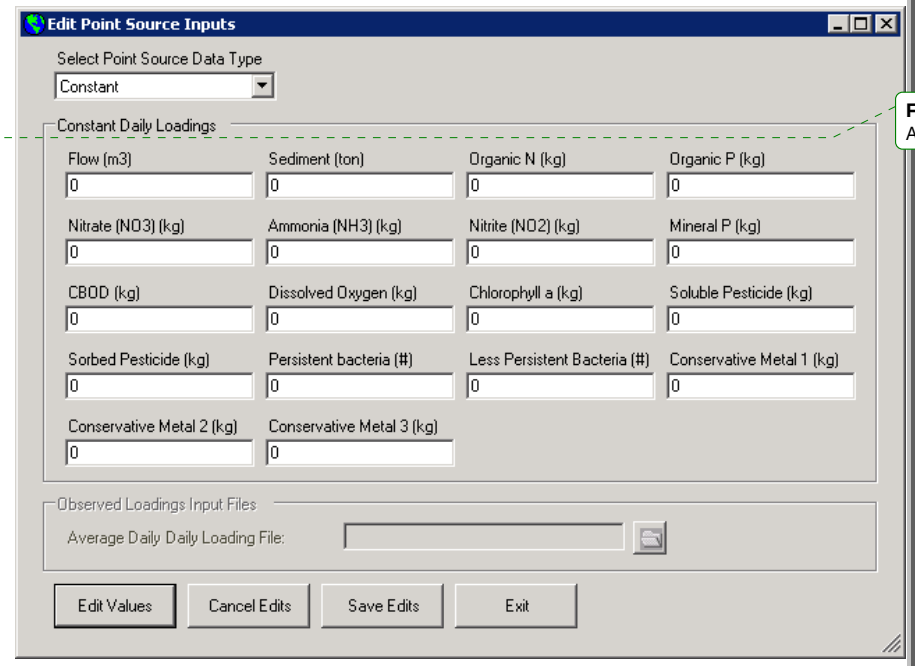


Figure 35 Edit Point Discharge Inputs Dialog

Formatted: Font: (Default) Arial

Formatted: French (France)

Field Code Changed

Formatted: French (France)

4. If you wish to edit the point sources in another subbasin, select it from the list in the “Edit Point Source Inputs” dialog box. Click **Exit** to complete editing of point discharges database in all subbasins.

Step 3: Edit Inlet Dischargers Input

1. Select the **Inlet Discharges** option under the **Edit Input** menu
2. If there are any inlet dischargers in the project, “Edit Inlet Input” dialog box with a list of subbasins containing inlet dischargers will appear (Figure 36).
3. You will be able to modify the inlet input information using a procedure similar to editing the point dischargers database
4. The dialog box allows the input of inlet discharge data in one of four formats: constant daily loadings, average annual loadings, average monthly loadings, and daily loadings. Choose a format by clicking the button next to the format to be used.
5. The default inlet discharge data format option is constant daily loadings. If you select this format, you will be prompted to input average daily flow (m^3), sediment loading (tons), and organic N, organic P, NO_3 and mineral P loadings.
6. If you choose “Annual Records”, “Monthly Records” or “Daily Records” option you will be prompted to load the data from the disk.

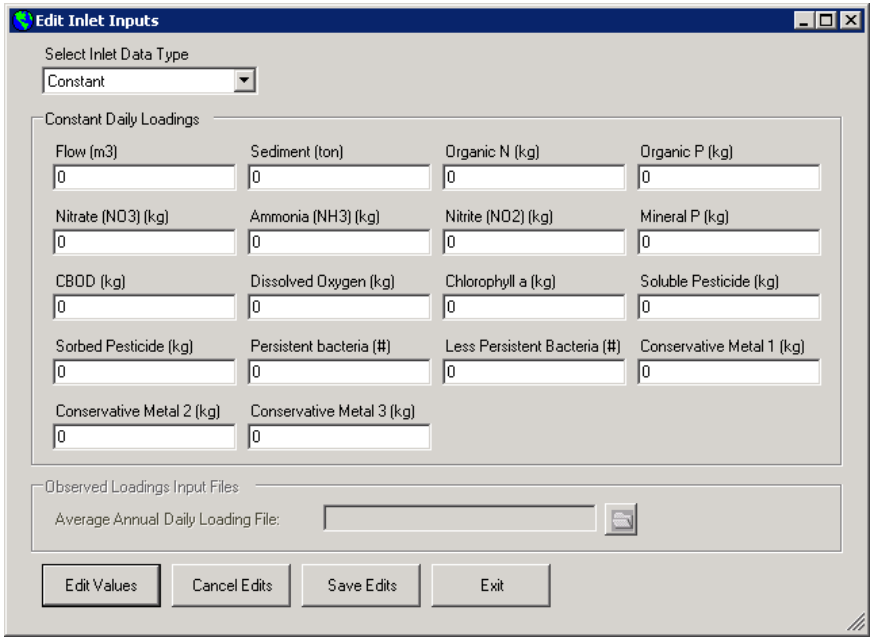


Figure 36 Edit Inlet Discharge Inputs Dialog

7. Click **OK** to complete the editing of inlet discharges for a subbasin.
8. If you wish to edit the inlets in another subbasin, select it from the list in the "Edit Inlet Discharger Input" dialog box. Click **Exit** to complete editing of inlet dischargers input in all subbasins
9. Since there are no inlet dischargers defined in this tutorial you will get a message "**No Inlet Discharges in the Watershed**"

Step 4: Edit Reservoir Input

1. To edit the reservoirs, on the **Edit Input** menu, select **Reservoirs**. A dialog box will appear with a list of the subbasins containing reservoirs.
2. To edit the reservoirs within a subbasin, click on the number of the subbasin in the "Edit Reservoirs Inputs" dialog box.
3. Since there are no reservoirs defined in this project you will get a message "**No Reservoirs in the Watershed**".

Step 5: Edit Subbasins Data

1. To edit the subbasin input files, select the **Subbasins Data** option under the **Edit Input** menu. "Edit Subbasin Inputs" dialog box will appear (Figure 37 Edit Subbasin Inputs main dialog).

   This dialog box contains the list of subbasins, land uses, soil types and slope levels within each subbasin and the input files corresponding to each subbasin/land use/soil/slope combination. To select an input file, select the subbasin, land use, soil type and slope that you would like to edit. When you select a subbasin, the combo box of land uses, soil types, and slope levels will be activated in sequence. Specify the subbasin/land use/soil combination of interest by selecting each category in the combo box.

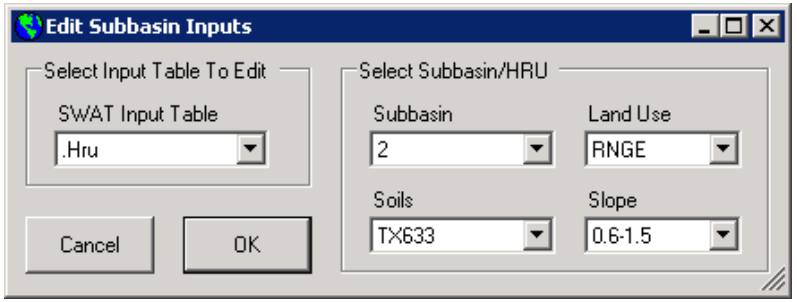


Figure 37 Edit Subbasin Inputs main dialog

2. To edit the soil physical data, click on the .sol extension, and select the subbasin number, landuse type, soil type and slope level. Then the OK button is activated. Click OK; a new dialog box will appear (Figure 38). Click the **Edit Values** button; all the boxes are activated and the user can revise the default values.

3. The interface allows the user to save the revision of current .sol file to other .sol files. Three options are available: 1) extend edits to current HRU, which is the default setting, 2) extend edits to all HRUs, or 3) extend edits to selected HRUs. For the third option, the user needs to specify the subbasin number, landuse type, soil type and slope levels for the HRUs that the user wants to apply current .sol file parameters.

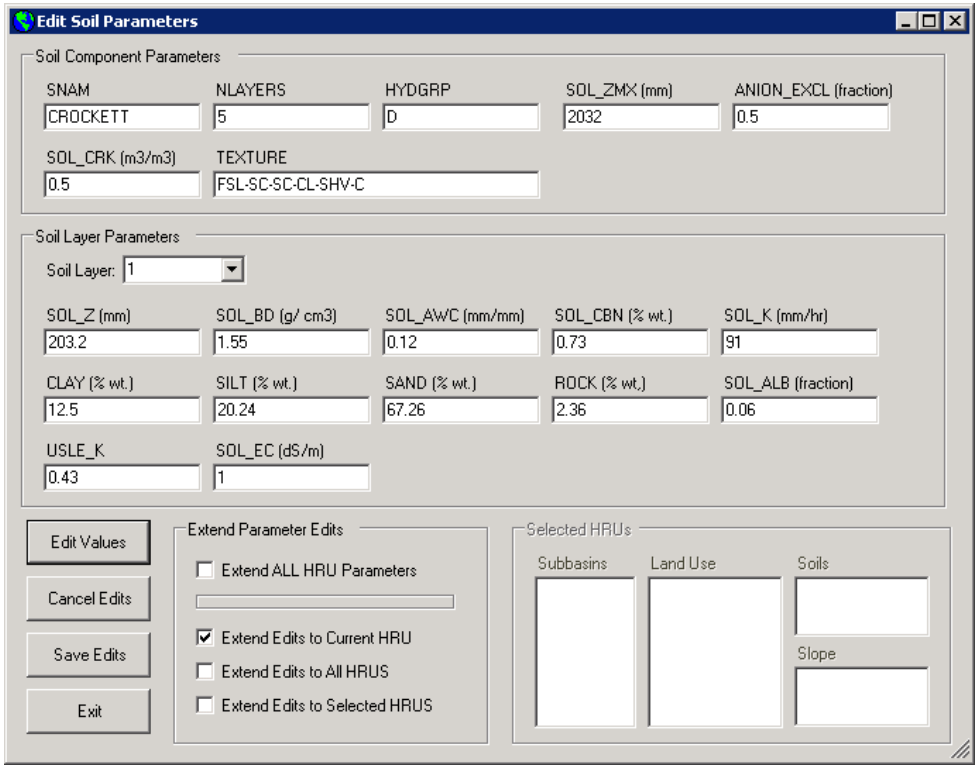


Figure 38 Edit soil input file dialog

4. To edit the weather generator data click on the .wgn extension. For the .wgn file you only need to select the subbasin number, and then the OK button will be activated. Click OK; a new dialog box (Figure 39) will appear which will allow you to modify the data in .wgn file. Similar to .sol file, the interface also allow the user to extend the current edits to other subbasins. The user can select to 1) extend edits to current Subbasin, which is the default setting, 2) extend edits to all Subbasins, or 3) extend edits to selected Subbasins.

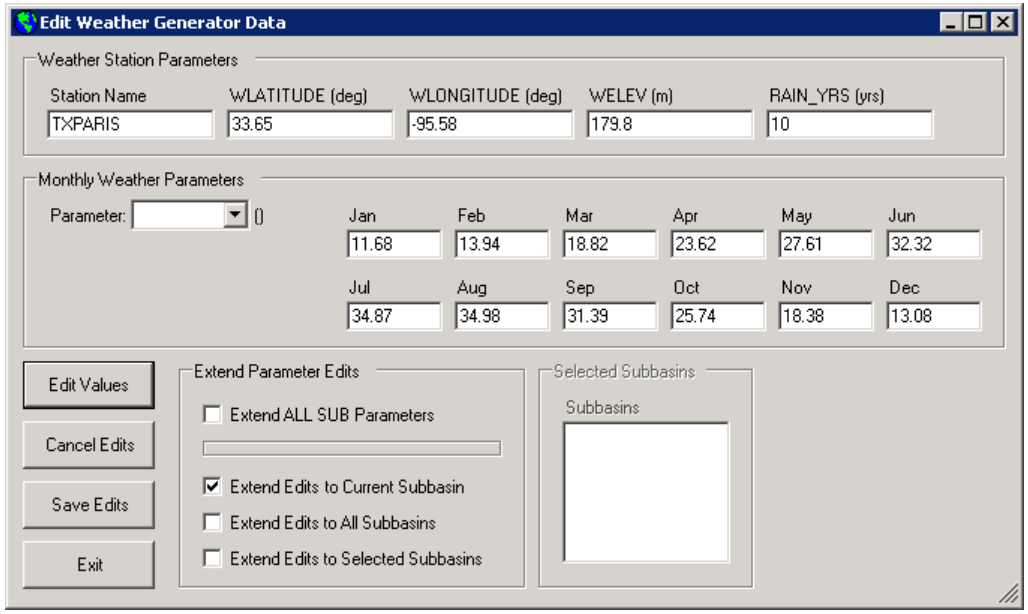


Figure 39 Edit weather generator input file main dialog

5. To edit general subbasin data, click on the .sub extension in the “Select Input File”. Select the subbasin number and click OK, then a new dialog box will appear displaying the existing general subbasin data for the selected subbasin (Figure 40). To modify data, activate all fields by clicking **Edit Values** button. For the elevation band parameters, the user can choose ELEVB, ELEVB_FR and SNOEB in the combo box beside the Elevation Band, then change the parameter values for each band. Also, the user can choose RFINC, TMPINC, RADINC, HUMINC in the combo box aside of Weather Adjustment, then change the parameter values for each month. Once you have made all editing changes, click the **OK** button. The interface will save all changes and return you to the “Edit Subbasin Inputs” box. The user can select to 1) extend edits to current Subbasin, which is the default setting, 2) extend edits to all Subbasins, or 3) extend edits to selected Subbasins. For selecting multiple subbasins, hold the Shift key when clicking the preferred subbasin numbers.

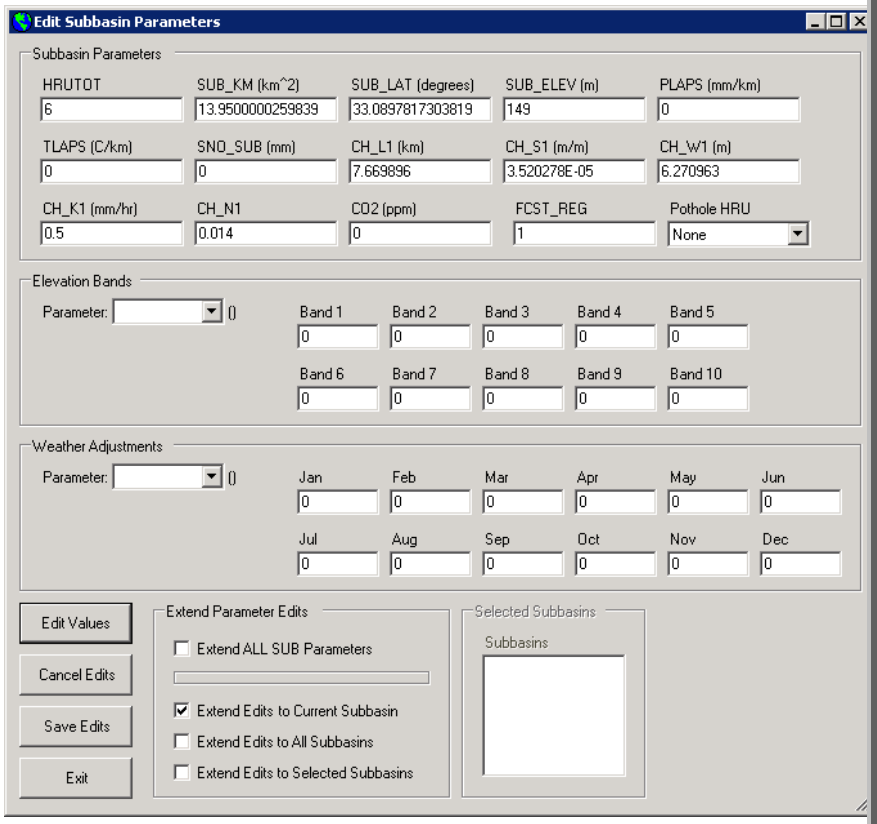


Figure 40 Edit Subbasin (Sub) dialog

6. To edit general HRU data, click on the .hru extension in the “Select Input File” section of the “Edit Subbasin Inputs” dialog box (Figure 41). A new dialog box will appear displaying the existing general HRU data for the selected subbasin. To modify data, activate a field by positioning the cursor over the text box and clicking. Once a cursor appears in the field, make the desired changes. Once you have made all editing changes, click the **OK** button. If you do not want to copy the edited HRU generator data to other data sets, click **No** on the prompt dialog.

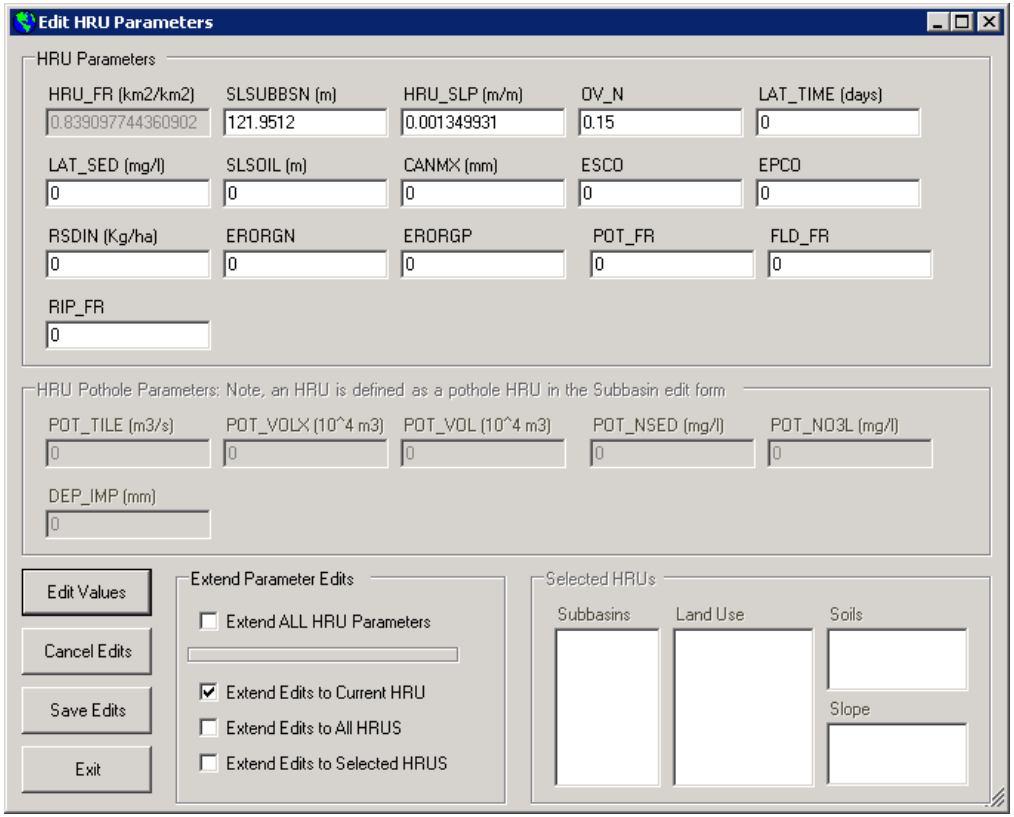


Figure 41 Edit Hydrologic Response Units (HRU) dialog.

7. To edit the main channel input file, click on the .rte in the “Select Input file” section of “Edit Subbasin Inputs” dialog box. A new dialog box (Figure 42) will appear with the existing main channel data for the selected subbasin. Click **Edit Values** button to activate all the textboxes to all user’s modification. Also the user can extend current edits to other basins with three types of options.

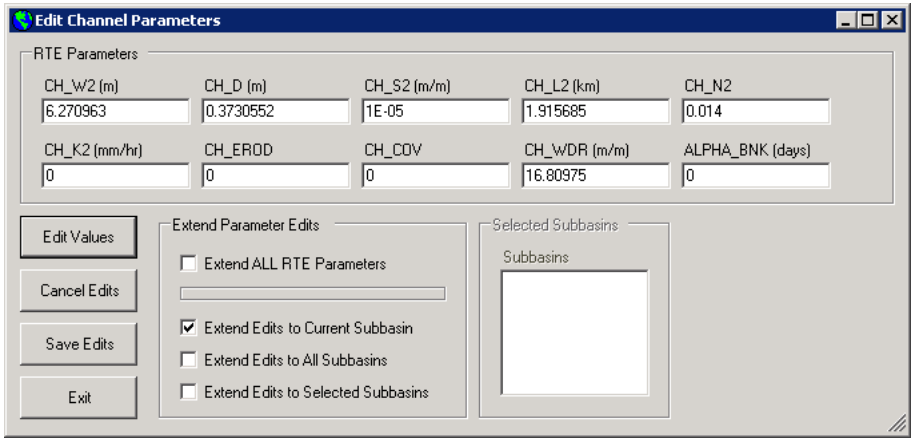


Figure 42 Edit Main channel input data dialog

8. To edit the ground water input file, click on the .gw in the “Select Input file” section of “Edit Subbasin Inputs” dialog box. In the dialog box that opens (Figure 43) with the existing data, make modifications by clicking **Edit Values** button to activating all textboxes. Similarly, after the modification, the user has three options to Save Edits to other HRUs.

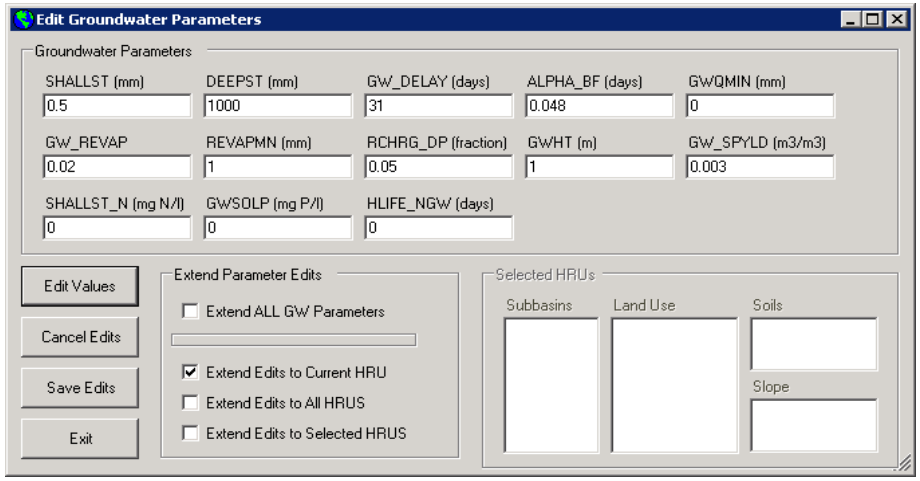


Figure 43 Edit Ground Water input data dialog

9. To edit the consumptive water use input data, click on the .wus in the “Select Input file” section of “Edit Subbasin Inputs” dialog box. In the dialog box (Figure 44) that opens with the existing data, click **Edit Values** button, then the user can modify the data. Also the current edits can be saved to other subbasins.

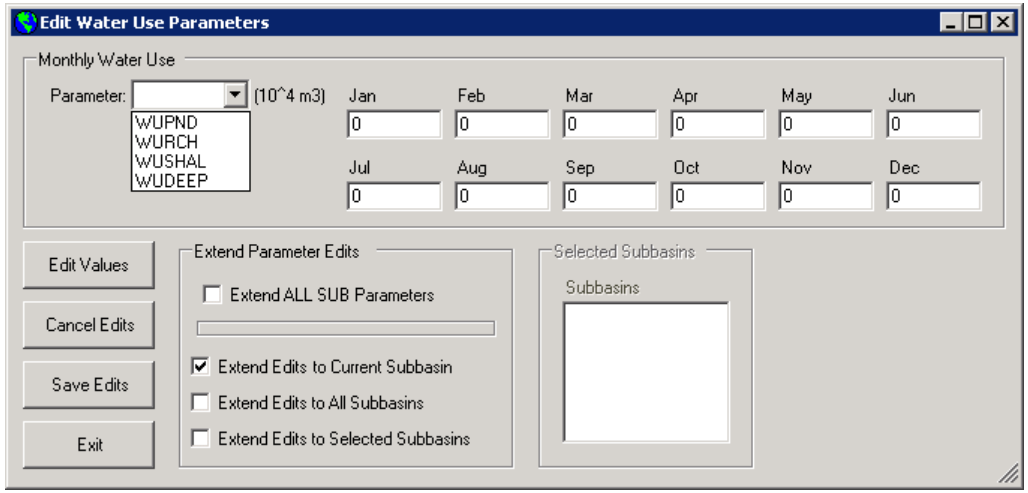


Figure 44 Edit water use input data dialog

10. To edit the management file input data, click on the .mgt in the “Select Input file” section of “Edit Subbasin Inputs” dialog box. A new dialog box (Figure 45) will appear and display the management data editor. This dialog has two tabs: General Parameters and Operations. In the first tab the user can modify the general parameters concerned with Initial Plant Growth, General Management, Urban Management, Irrigation Management, and Tile Darin Management. In the second tab, the user can arrange the detailed management options on the current HRU. The management operations can be scheduled by Date or by Heat Units. The settings of the management operations can also be extended to other HRUs that the user has defined.

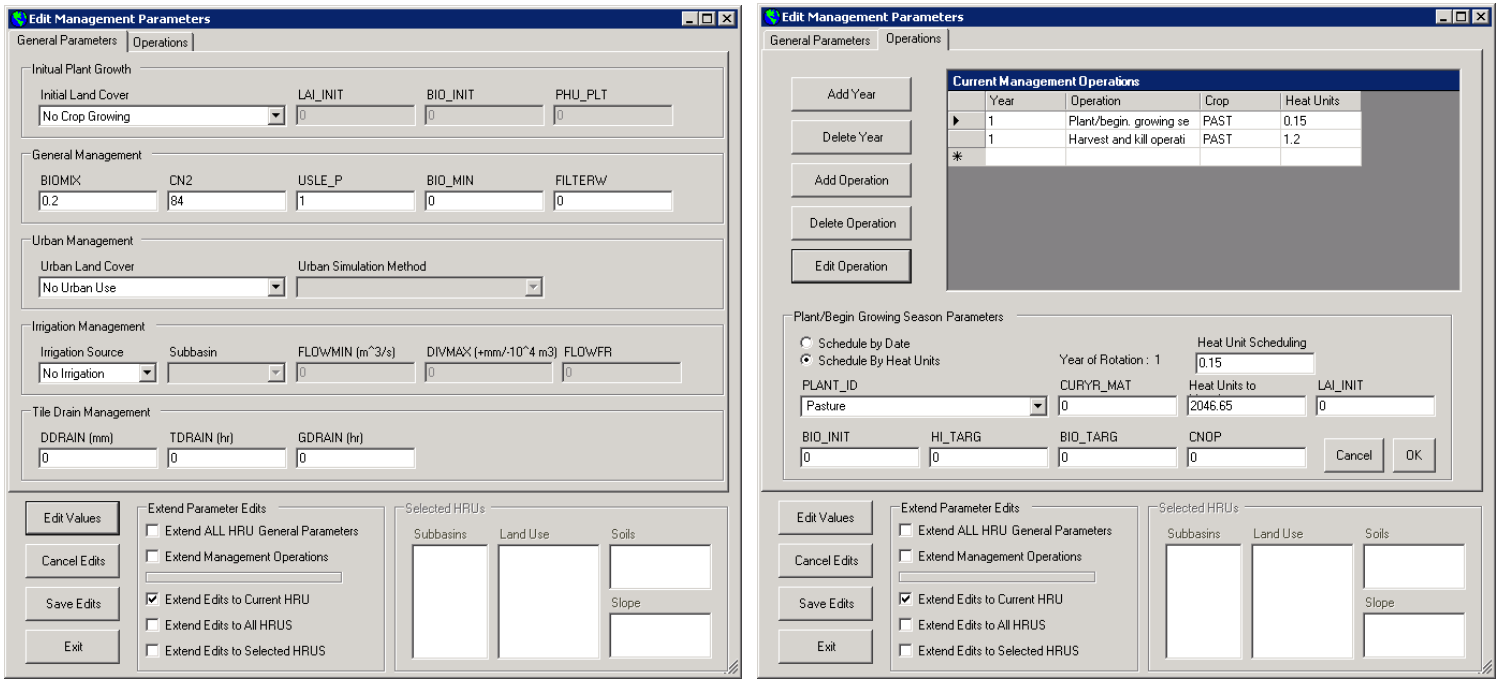


Figure 45 Edit management input file main dialog

11. To edit the soil chemical data click on the .chm in the “Select Input file” section of “Edit Subbasin Inputs” dialog box. A new dialog box (Figure 46) will appear displaying the Soil Chemical data editor. To modify the displayed data, click the **Edit Values** button to activate all the textboxes. After the modification of Soil Chemical Data, the user also can extend the modification to other user specified HRUs.

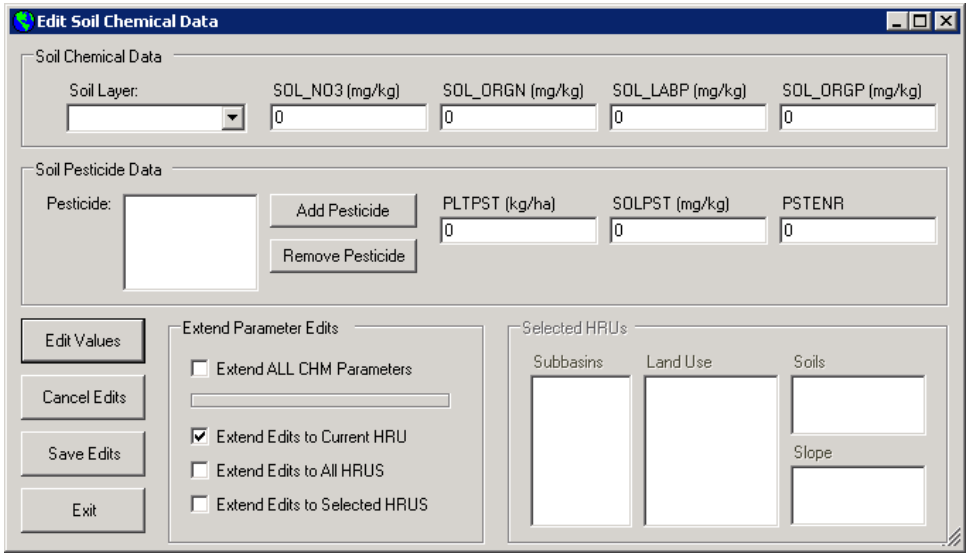


Figure 46 Soil chemical input data editor

12. To edit pond data click on the .pnd in the “Select Input file” section of “Edit Subbasin Inputs” dialog box. A new dialog box (Figure 47) will appear displaying the pond data editor. To modify the displayed data, click the **Edit Values** button to activate all the textboxes. After the modification of pond data, the user also can extend the modification to other user specified Subbasins.

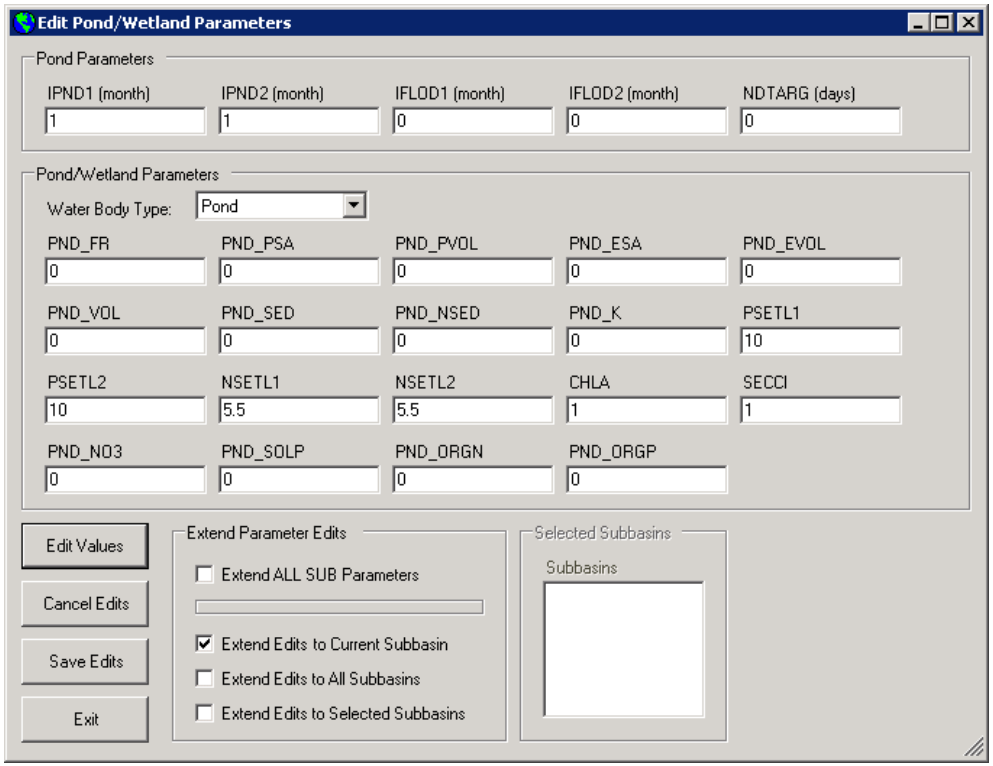


Figure 47 Dialog to edit pond input data file

13. To edit stream water quality input data file click on the .swq in the “Select Input file” section of “Edit Subbasin Inputs” dialog box. A new dialog box (Figure 48) will appear displaying the stream water quality input data editor. To modify the displayed data, click the **Edit Values** button to activate all the textboxes. After the modification of stream water quality input data, the user also can extend the modification to other user specified Subbasins.

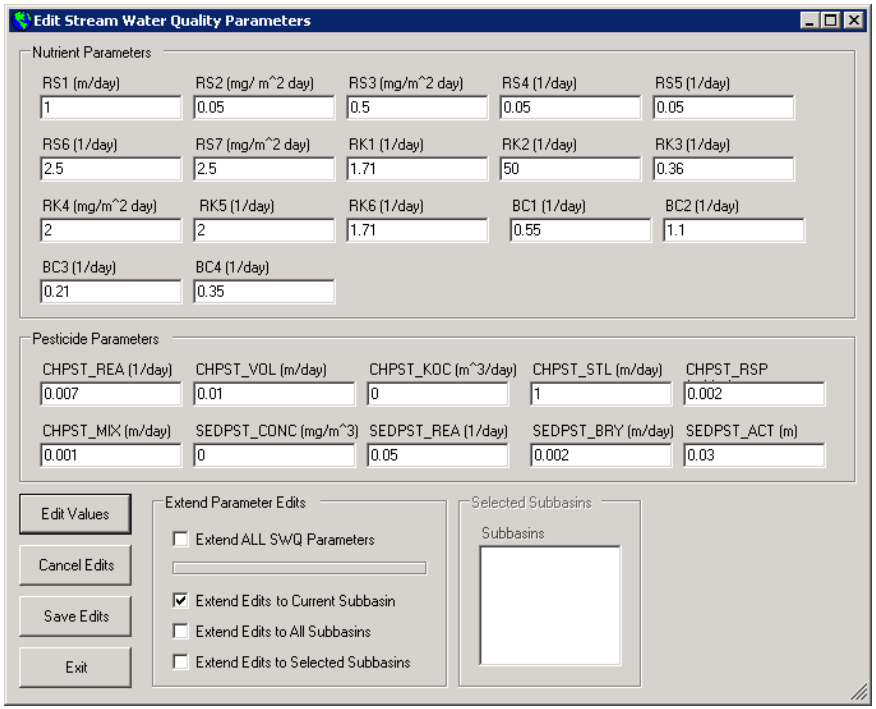


Figure 48 Stream water quality input data editor

14. Go to the **Watershed Data** item under the **Edit SWAT Input** menu, and click the General Data (.BSN) button, then a new dialog (Figure 49) will appear. This interface allows you to modify the parameters concerned with three major groups: 1) Water Balance, Surface Runoff, and Reaches, 2) Nutrients and Water Quality, and 3) Basin-wide Management. After revision of the parameters, click Save Edits.

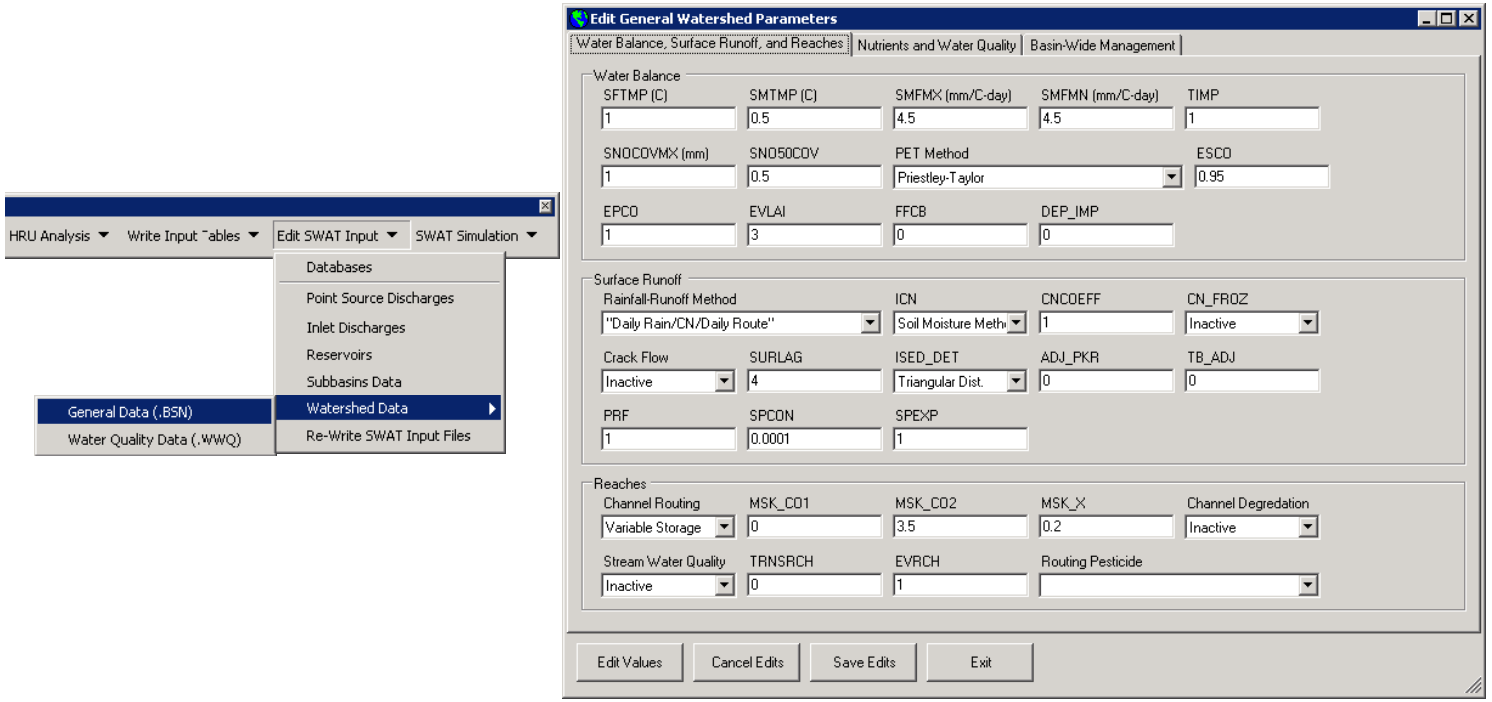


Figure 49 General Watershed Parameters Editor

15. Go to the **Watershed Data** item under the **Edit SWAT Input** menu, and click the General Data (.BSN) button, then a new dialog (Figure 50) will appear. This interface allows you to modify the parameters concerned with Watershed Water Quality Simulation. After revision of the parameters, click Save Edits.

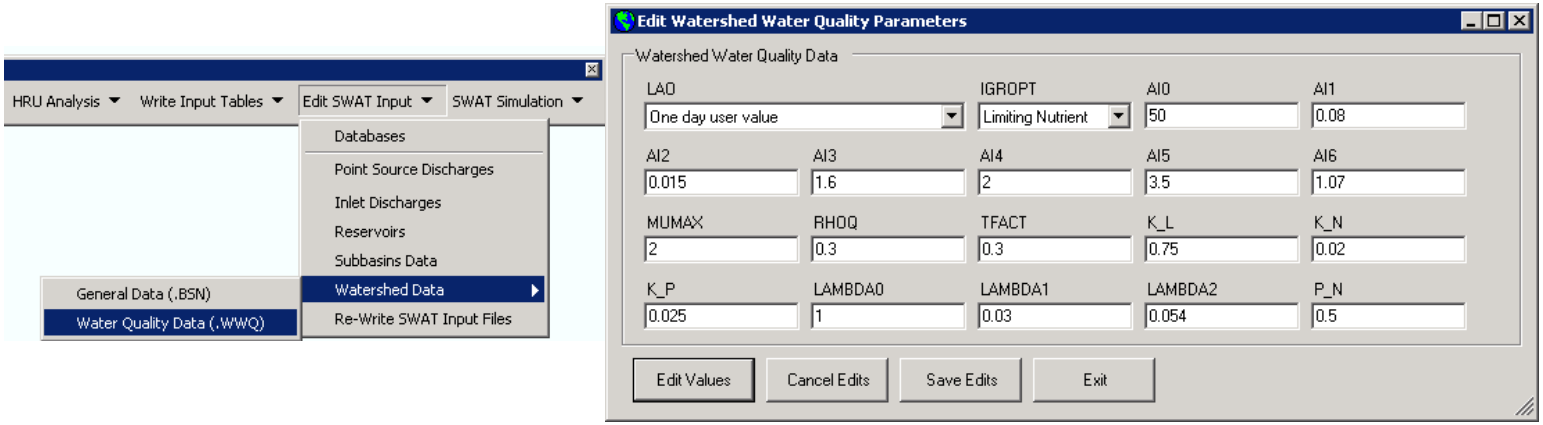


Figure 50 Watershed Water Quality Parameters

SWAT Simulation Setup

SWAT simulation menu contains commands that setup and run SWAT simulation. To build SWAT input files and run the simulation, proceed as follows:

Step 1: Setup data and Run SWAT

1. Select the **Run SWAT** command under the **SWAT Simulation** menu. It will open a dialog box (Figure 51) that will allow you to set up the data for SWAT simulation.

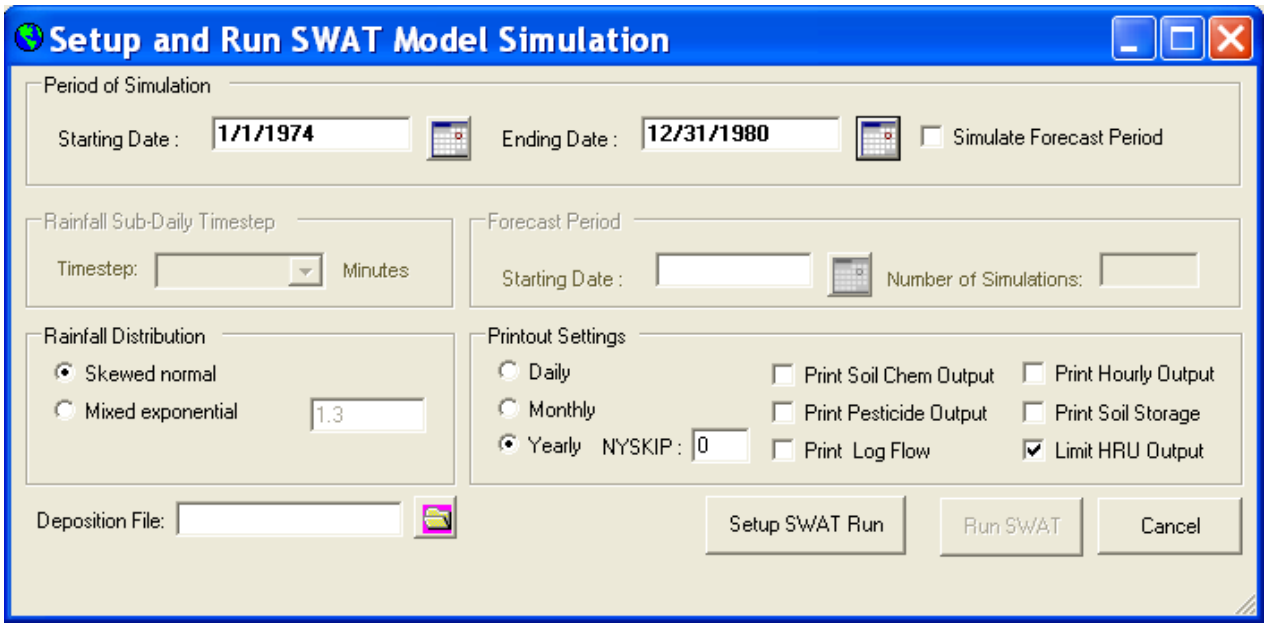


Figure 51 SWAT data setup and simulation dialog

2. Select the **1/1/1974** for the "**Starting date**" and **12/31/1980** for the "**Ending date**" option. If you are using simulated rainfall and temperature data, both these fields will be blank and you have to input the information manually.
3. Choose "monthly" option for **Printout Frequency**
4. Keep the rest at the default selections.
5. After all the parameters have been set, click the **Setup SWAT Run** button in the "Run SWAT" dialog box (Figure 51) to build the SWAT CIO, COD, PCP.PCP and TMP.TMP input files. Once all input files are setup, the **Run SWAT** button is activated in the bottom right of the Run SWAT dialog.
6. Click the button labeled **Run SWAT**. This will run the SWAT executable file. A message box will indicate the successful completion of SWAT run.
7. The button **Save SWAT Run** is activated now. You need to save the current setting of the SWAT project to another folder. If the user doesn't use this function, the user interface will not allow the user to use the "**Sensitivity Analysis**" and "**Auto-calibration and Uncertainty Analysis**" functions. Click the **Save SWAT Run** button, and input "Sim1" as the name of current SWAT Run. Click OK. Then the interface will copy the files under **"D:\SWAT\Nzoia\Scenarios\Default**" to **"D:\SWAT\Nzoia\Scenarios\Default**". And a dialog will appear to notify you that the current SWAT run has been saved as Sim1 (Figure 52).

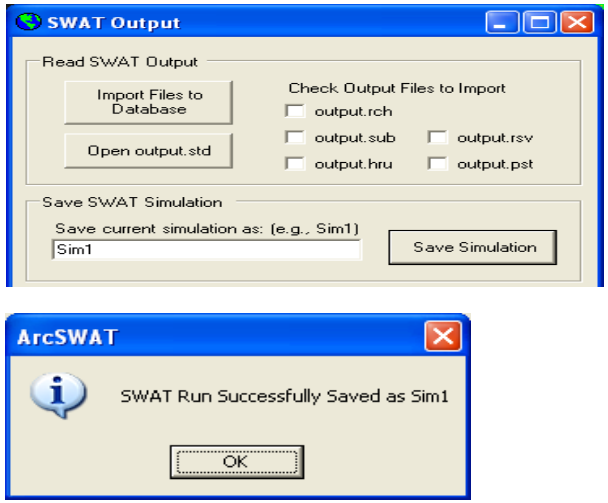
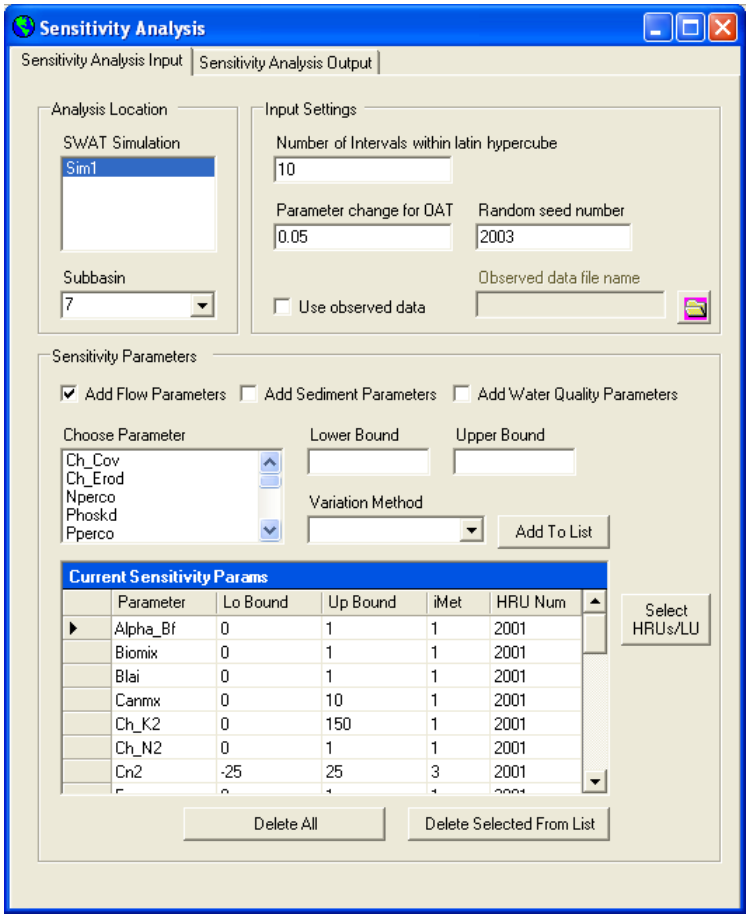


Figure 52 Dialog of saving SWAT run

Step 2: Sensitivity Analysis of SWAT

After saving current SWAT run to another folder, the buttons **Sensitivity Analysis** and **Auto Calibration and Uncertainty Analysis** are activated under the **SWAT Simulation** menu.

1. Click **Sensitivity Analysis** button. A dialog for sensitivity analysis of SWAT will appear (Figure 53). In order to run sensitivity analysis of SWAT, the name of SWAT run and the location of subbasin needs to be specified.
2. Click the "Sim1" under the SWAT simulation, select a subbasin to run analysis on, and check the appropriate Sensitivity Parameters. Click the **Write Input Files** button on the bottom of the dialog box. A folder named "Sensitivity" will be created under the folder D:\SWAT\Nzoia\Scenarios\Default\TxtInOut. After writing the input files the **Run Sensitivity Analysis** button is activated. Clicking this button executes the SWAT2005.exe file.


Sensitivity Analysis
Sensitivity Analysis Input
Sensitivity Analysis Output
Analysis Location
SWAT Simulation
Sim1
Subbasin
7
Input Settings
Number of Intervals within latin hypercube
10
Parameter change for OAT
0.05
Random seed number
2003
Observed data file name
Use observed data
Sensitivity Parameters
Add Flow Parameters
Add Sediment Parameters
Add Water Quality Parameters
Choose Parameter
Ch_Cov
Ch_Erod
Nperco
Phoskd
Pperco
Lower Bound
Upper Bound
Variation Method
Add To List
Current Sensitivity Params
Parameter
Lo Bound
Up Bound
iMet
HRU Num
Alpha_Bf 0 1 1 2001
Biomix 0 1 1 2001
Blai 0 1 1 2001
Canmx 0 10 1 2001
Ch_K2 0 150 1 2001
Ch_N2 0 1 1 2001
Cn2 -25 25 3 2001
Select HRUs/LU
Delete All
Delete Selected From List

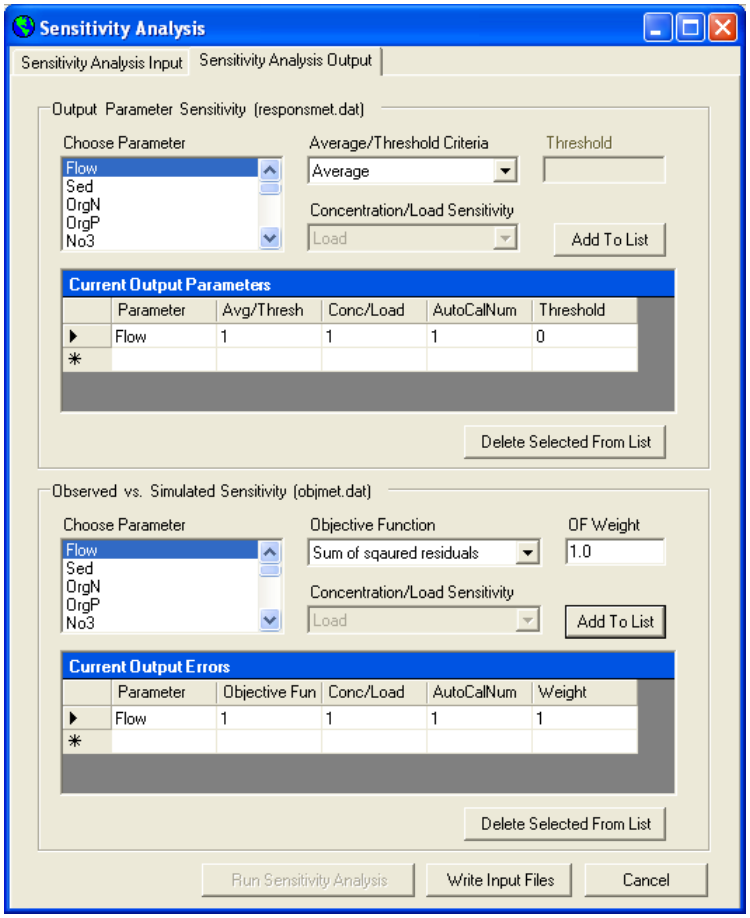


Figure 53 Dialog for sensitivity analysis of SWAT

Appendix: Installing ArcSWAT

System Requirements

The SWAT2005/ArcSWAT 1.0 Interface requires:

Hardware:

- Personal computer using a Pentium IV processor or higher, which runs at 2 gigahertz or faster
- 1 GB RAM minimum
- 500 megabytes free memory on the hard drive for minimal installation and up to 1.25 gigabyte for a full installation (including sample datasets and US STATSGO data)

Software (ArcSWAT 1.0 for ArcGIS 9.1 version):

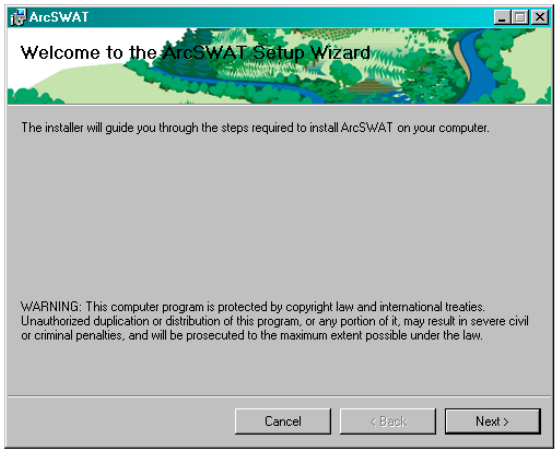
- Microsoft Windows XP, or Windows 2000 operating system with most recent kernel patch*****
- ArcGIS-ArcView 9.1 with service pack 2 (Build 766)
- ArcGIS Spatial Analyst 9.1 extension
- ArcGIS Developer Kit (usually found in C:\Program Files\ArcGIS\DeveloperKit\)
- ArcGIS DotNet support (usually found in C:\Program Files\ArcGIS\DotNet\)
- Microsoft .Net Framework 1.1
- Adobe Acrobat Reader version 7 or higher

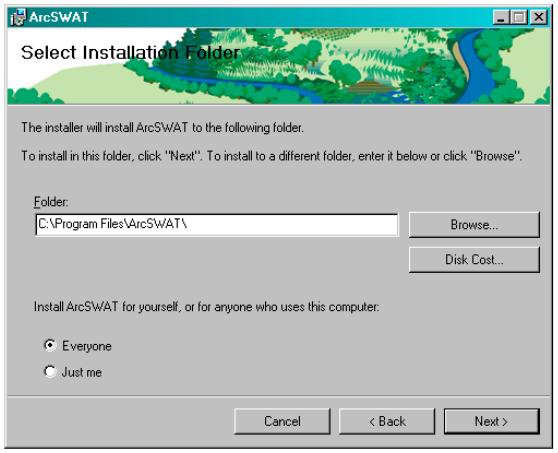
Microsoft constantly updates the different versions of windows. This interface was developed with the latest version of Windows and may not run with earlier versions. Patches are available from Microsoft.

Using the ArcSWAT Setup Wizard:

After downloading the ArcSWAT program, open the ArcSWAT_Install_1.0.0 folder. Click the 

icon to begin installation. Follow the installation wizard instructions.

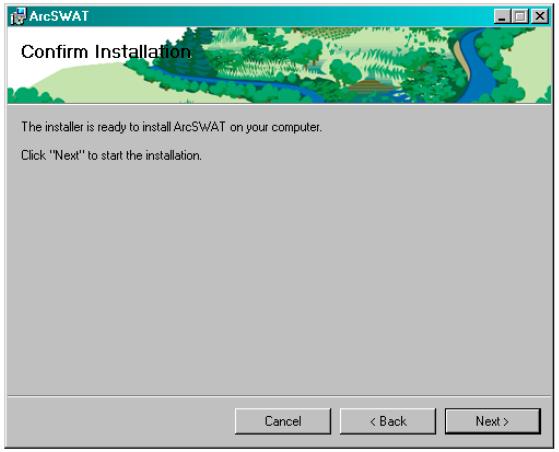


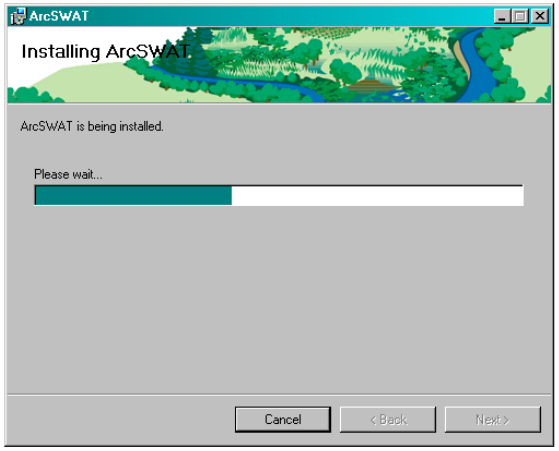


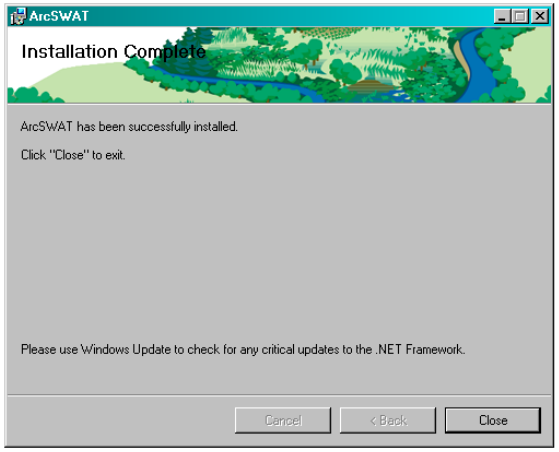
Select the appropriate folder location for the program, preferably the computer's main hard drive. Click the **Disk Cost** button to ensure enough disk space for installation.

Indicate if program access will be for everyone who uses the computer or just the installer.

Follow the wizard instructions until installation is completed successfully.







Additional information on ArcSWAT installation under ArcGIS 9.1:

What are the build numbers for all the recent releases of ArcGIS?

http://support.esri.com/index.cfm?fa=knowledgebase.techArticles.articleShow&d=30104

You need ArcGIS Desktop 9.1 Service Pack 2:

http://support.esri.com/index.cfm?fa=downloads.patchesServicePacks.viewPatch&PID=17&MetaID=1162

How to install ArcGIS 9.1 with .NET support?

1. Insert the ArcView or ArcGIS Desktop installation disk.

2. Select Install ArcGIS Desktop.

3. Select Modify.

4. Expand Applications; verify that '.NET Support' is installed. If you see a red X, click on the X and select 'Entire feature will be installed' and then follow the rest of the wizard.